

OLYMPUS[®]

User Manual

cellSens

LIFE SCIENCE IMAGING SOFTWARE

Any copyrights relating to this manual shall belong to OLYMPUS CORPORATION.

We at OLYMPUS CORPORATION have tried to make the information contained in this manual as accurate and reliable as possible. Nevertheless, OLYMPUS CORPORATION disclaims any warranty of any kind, whether expressed or implied, as to any matter whatsoever relating to this manual, including without limitation the merchantability or fitness for any particular purpose. OLYMPUS CORPORATION will from time to time revise the software described in this manual and reserves the right to make such changes without obligation to notify the purchaser. In no event shall OLYMPUS CORPORATION be liable for any indirect, special, incidental, or consequential damages arising out of purchase or use of this manual or the information contained herein.

No part of this document may be reproduced or transmitted in any form or by any means, electronic or mechanical for any purpose, without the prior written permission of OLYMPUS CORPORATION.

All brands are Trademark or registered Trademark of their respective owners.

© OLYMPUS CORPORATION
All rights reserved

Version 510_UMA_cellSens1.13-Nile-en_00_24February2015

Contents

1. About the documentation for your software	5
2. Overview of the user interface	6
2.1. Overview - User interface	6
2.2. Overview - Layouts	7
2.3. Document group	8
2.4. Tool Windows	10
2.5. Image window views	12
2.6. Working with documents	13
3. Configuring the system	16
4. Acquiring a single image	18
4.1. Snapshot	18
4.2. Behavior of the live window	19
4.3. Acquiring HDR images	21
5. Acquiring multi-dimensional images	24
5.1. Overview - Acquisition processes	24
6. Acquiring image series	27
6.1. Time stacks	27
6.2. Z-stack	33
7. Acquiring fluorescence images	36
7.1. Before and after you've acquired a fluorescence image	39
7.2. Defining observation methods for the fluorescence acquisition	41
7.3. Acquiring and combining fluorescence image	47
7.3.1. Acquiring individual fluorescence images	47
7.3.2. Combine Channels	48
7.4. Acquiring multi-channel fluorescence images	53
8. Creating stitched images	59
9. Processing images	66
10. Life Science Applications	67
10.1. Intensity profile	68
10.2. Fluorescence Unmixing	74
10.2.1. Carrying out a fluorescence unmixing	75
10.3. Colocalization	79
10.3.1. Overview	79
10.3.2. Measuring the colocalization	80
10.4. Deconvolution	85

10.5.	Ratio Analysis	87
10.5.1.	Overview	87
10.5.2.	Carrying out a Ratio Analysis	89
10.6.	Kymograph.....	94
10.6.1.	Working with the kymograph	95
11.	Measuring images	98
11.1.	Counting objects	98
11.2.	Using interactive measurement functions.....	102
11.2.1.	Overview	102
11.2.2.	Measuring images	105
11.3.	Carrying out an automatic image analysis.....	111
11.3.1.	Counting objects.....	111
11.3.2.	Counting object belonging to different phases (setting the thresholds)	113
11.3.3.	Measuring objects (selecting and outputting measurement parameters)	115
11.3.4.	Filtering objects	116
11.3.5.	Classifying objects.....	118
12.	Running experiments	120
12.1.	Overview	120
12.2.	Toolbar - Experiment plan	125
12.3.	Working with the Experiment Manager.....	127
12.3.1.	Defining a new experiment.....	127
12.3.2.	Using image analysis functions in an experiment	143
12.3.3.	Adapting existing experiments	149
13.	Working with reports	150
13.1.	Working with the report composer	151
13.2.	Working with the Olympus MS-Word add-in.....	155
13.2.1.	Creating and editing a page template	156
13.2.2.	Editing a report	159

1. About the documentation for your software

The documentation for your software consists of several parts: the installation manual, the online help, and PDF manuals which were installed together with your software.

Where do you find which information?

The installation manual is delivered with your software. There, you can find the system requirements. Additionally, you can find out how to install and configure your software.

In the manual, you can find both an introduction to the product and an explanation of the user interface. By using the extensive step-by-step instructions you can quickly learn the most important procedures for using this software.

In the online help, you can find detailed help for all elements of your software. An individual help topic is available for every command, every toolbar, every tool window and every dialog box.

New users are advised to use the manual to introduce themselves to the product and to use the online help for more detailed questions at a later date.

Writing convention used in the documentation

In this documentation, the term "your software" will be used for cellSens.

00054

Example images

The DVD that comes with your software contains, among a lot of other data, also images that show different examples of use for your software. You can load these so-called example images from the DVD. However, in many cases, installing the example images on your local hard disk or on a network drive is more helpful. Then the example images will always be available, no matter where the DVD with the software currently is.

Note: Your software's user documentation often refers to these example images. You can directly follow some step-by-step instructions when you load the corresponding example image.

You can open and view the example images with your software. Additionally, you can use the example images to test some of your software's functions, for example, the automatic image analysis, the image processing or the report creation.

Due to the fact that the example images also contain multi-dimensional images like Z-stacks or time-lapse images, making use of them enables you to quickly load images that require more complex acquisition settings.

Installing example images

You can install the example images after you've installed the software, or at any later point in time.

To do so, insert the DVD that contains the software into the DVD drive. If the installation wizard starts, browse to the directory that contains the example images and install them.

07005

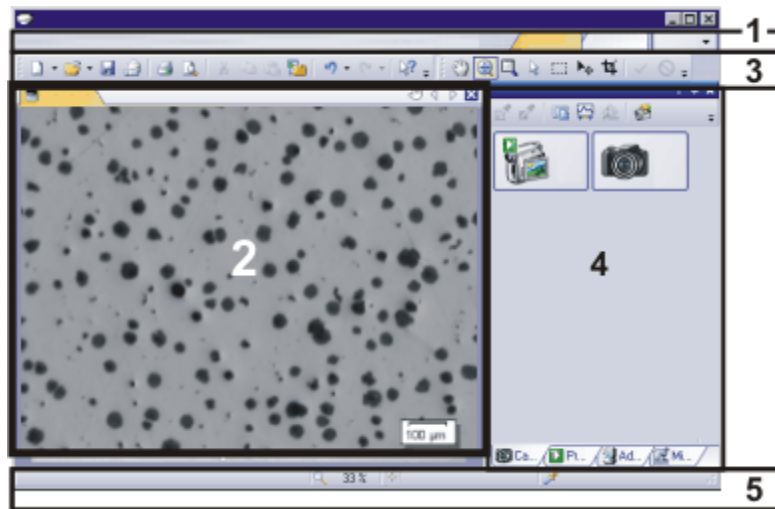
2. Overview of the user interface

2.1. Overview - User interface

The graphical user interface determines your software's appearance. It specifies which menus there are, how the individual functions can be called up, how and where data, e.g. images, is displayed, and much more. In the following, the basic elements of the user interface are described.

Note: Your software's user interface can be adapted to suit the requirements of individual users and tasks. You can, e.g., configure the toolbars, create new layouts, or modify the document group in such a way that several images can be displayed at the same time.

Appearance of the user interface



The illustration shows the schematic user interface with its basic elements.

- (1) *Menu bar* You can call up many commands by using the corresponding menu. Your software's menu bar can be configured to suit your requirements. Use the [Tools > Customization > Start Customize Mode...](#) command to add menus, modify, or delete them.
- (2) *Document group* The document group contains all loaded documents. These can be of all supported document types. When you start your software, the document group is empty. While you use your software it gets filled - e.g., when you load or acquire images, or perform various image processing operations to change the source image and create a new one.
- (3) *Toolbars* Commands you use frequently are linked to a button providing you with quick and easy access to these functions. Please note, that there are many functions which are only accessible via a toolbar, e.g., the drawing functions required for annotating an image. Use the [Tools > Customization > Start Customize Mode...](#) command to modify a toolbar's appearance to suit your requirements.
- (4) *Tool windows* Tool windows combine functions into groups. These may be very different functions. For example, in the [Properties](#) tool window you will find all the information available on the active document. In contrast to dialog boxes, tool windows remain visible on the user interface as long as they are switched on. That gives you access to the settings in the tool windows at any time.
- (5) *Status bar* The status bar contains a large amount of information, e.g., a brief description of each function. Simply move the mouse pointer over the command or button for this information.

2.2. Overview - Layouts

To toggle between different layouts, click on the right-hand side in the menu bar on the name of the layout you want, or use the [View > Layout](#) command.

Which predefined layouts are there?

For important tasks several layouts have already been defined. The following layouts are available:

- Acquire images ("Acquisition" layout)
- View and process images ("Processing" layout)
- Measuring images ("Count and Measure" layout)
- Generate a report ("Reporting" layout)

In contrast to your own layouts, predefined layouts can't be deleted. Therefore, you can always restore a predefined layout back to its originally defined form. To do this, select the predefined layout, and use the [View > Layout > Reset Current Layout](#) command.

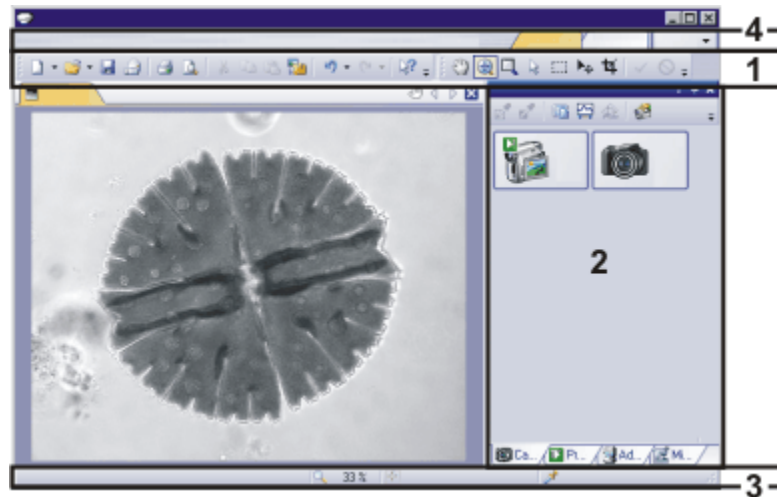
What is a layout?

Your software's user interface is to a great extent configurable, so that it can easily be adapted to meet the requirements of individual users or of different tasks. You can define a so-called "layout" that is suitable for the task on hand. A "layout" is an arrangement of the control elements on your monitor that is optimal for the task on hand. In any layout, only the software functions that are important in respect to this layout will be available.

Example: The [Camera Control](#) tool window is only of importance when you acquire images. When instead of that, you want to measure images, you don't need that tool window.

That's why the "Acquisition" layout contains the [Camera Control](#) tool window, while in the "Count and Measure" layout it's hidden.

Which elements of the user interface belong to the layout?



The illustration shows you the elements of the user interface that belong to the layout. The layout saves the element's size and position, regardless of whether they have been shown or hidden. When, for example, you have brought the [Windows](#) toolbar into a layout, it will only be available for this one layout.

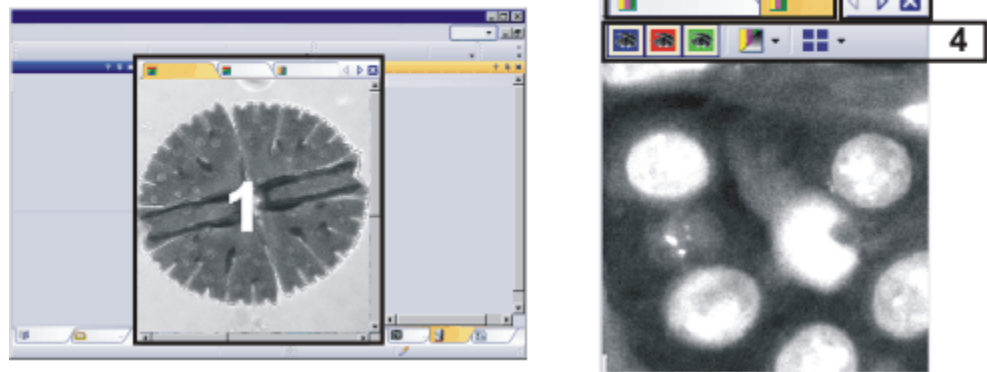
- (1) Toolbars
- (2) Tool windows
- (3) Status bar
- (4) Menu bar

00013

2.3. Document group

The document group contains all loaded documents. These can be of all supported document types. As a rule, the loaded documents are images.

Appearance of the document group



- (1) Document group in the user interface
- (2) Document bar in the document group
- (3) Buttons in the document bar
- (4) Navigation bar in the image window

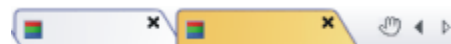
Document group in the user interface

You will find the document group in the middle of the user interface. In it you will find all of the documents that have been loaded, and of course also all of the images that have been acquired. Also the live-image and the images resulting from, e.g., any image processing function, will be displayed there.

Note: At the same time, up to 150 documents can be loaded in the document group.

Document bar in the document group

The document bar is the document group's header.



For every loaded document, an individual tab showing the document name will be set up in the document group. Click the name of a document in the document bar to have this document displayed in the document group. The name of the active document will be shown in color. Each type of document is identified by its own icon.

At the top right of each tab, a small [x] button is located. Click the button with the cross to close the document. If it has not yet been saved, the *Unsaved Documents* dialog box will open. You can then decide whether or not you still need the data.

Buttons in the document bar

At the top right of the document bar you will see several buttons.



 *Button with a hand*

Click the button with a hand on it to extract the document group from the user interface. In this way you will create a document window that you can freely position or change in size.

If you would like to merge two document groups, click the button with the hand in one of the two document groups. With the left mouse button depressed, drag the document group with all the files loaded in it, onto an existing one.

Prerequisite: You can only position document groups as you wish when you are in the expert mode. In standard mode the button with the hand is not available.

 *Arrow buttons*

The arrow buttons located at the top right of the document group are, to begin with, inactive when you start your software. The arrow buttons will only become active when you have loaded so many documents that all of their names can no longer be displayed in the document group. Then you can click one of the two arrows to make the fields with the document names scroll to the left or the right. That will enable you to see the documents that were previously not shown.

Navigation bar in the image window

Multi-dimensional images have their own navigation bar directly in the image window. Use this navigation bar to set or to change how a multi-dimensional image is to be displayed on your monitor.

There are some other document types with their own navigation bar directly in the image window. One example is a report instruction or an experiment plan.

00139

2.4. Tool Windows

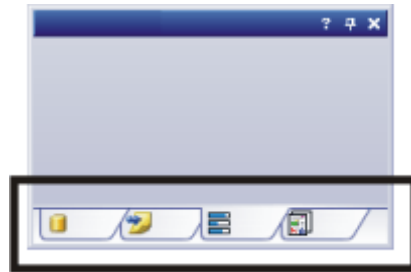
Tool windows combine functions into groups. These may be very different functions. For example, in the *Properties* tool window you will find all the information available on the active document.

Position of the tool windows

The user interface is to a large degree configurable. For this reason, tool windows can be docked, freely positioned, or integrated in document groups.

Docked tool windows

Tool windows can be docked to the left or right of the document window, or below it. To save space, several tool windows may lie on top of each other. They are then arranged as tabs. In this case, activate the required tool window by clicking the title of the corresponding tab below the window.



Freely positioned tool windows

You can only position tool windows as you wish when you are in the expert mode.

You can at any time float a tool window. The tool window then behaves exactly the way a dialog box does. To release a tool window from its docked position, click on its header with your left mouse button. Then, while pressing the left mouse button, drag the tool window to wherever you want it.

Integrating a tool window into a document group

You can only position tool windows as you wish when you are in the expert mode.

You can integrate certain tool windows in the document group, for example, the *File Explorer* tool window. To do this, use the *Document Mode* command. To open a context menu containing this command, rightclick any tool window's header.

The tool window will then act similarly to a document window, e.g., like an image window.

Use the *Tool Window Mode* command to float a tool window back out of the document group. To open a context menu containing this command, rightclick any tool window's header.

Buttons in the header

In the header of every tool window, you will find the three buttons *Help*, *Enable Auto Hide*, and *Close*.



Click the *Help* button to open the online help for the tool window.

Click the *Enable Auto Hide* button to minimize the tool window.

Click the *Close* button to hide the tool window. You can make it reappear at any time, for example, with the *View > Tool Windows* command.

Context menu of the header

To open a context menu, rightclick a tool window's header. The context menu can contain the *Enable Auto Hide*, *Document Mode* and *Transparency* commands. Which commands will be shown, depends on the tool window.

Additionally, the context menu contains a list of all of the tool windows that are available. Every tool window is identified by its own icon. The icons of the currently displayed tool windows will appear clicked. You can recognize this status by the icon's background color.

Use this list to make tool windows appear.

00037

2.5. Image window views

When working with multi-dimensional images you can choose between different views of the image in the image window. Multi-dimensional images have their own navigation bar directly in the image window. Click the small arrow next to the last button on this navigation bar to open a menu with commands you can use with image window views. In it, you can select the image window view you want and also edit the settings for some views.



This button is configured in such a way that you can switch backwards and forwards exactly between two different views simply by clicking it once.

Click the button to switch to the image window view that is currently shown as an icon on the button. Every image window view has its own icon.

The button always shows the image window view that was previously selected. For example, when you switch from the single frame view to the *Slice View* image window view, the button will automatically change its appearance to show the icon for the single frame view, making it possible for you to immediately switch back to that view.



By default you will find yourself in the single view. In the single frame view, only one image will be shown in the image window.



Use the tile view to attain an overview of all of the individual images that make up a multi-dimensional image. In this view, you can also select individual images.



Use the *Slice View* image window view, to look at any cross sections of an image series you want. The *Slice View* tool window offers you numerous possibilities for configuring this view.



You can display a Z-stack as a 3D object. To do so, use the *Voxel View* image window view, and the *Voxel View* tool window.

Projection Views

For image series, e.g. Z-stacks and time stacks, a single projection image can be calculated from all of the frames that is representative for the whole multi-dimensional image. The available projection images differ in the calculation algorithm. For example, if you use the maximum intensity projection you will, from all frames, only see the pixels with the highest intensity values.

EFI Projection

For Z-stacks an EFI projection is available. The EFI projection uses a series of differently focused separate images ("Focus series") to calculate a resulting image ("EFI image"), that is focused in all of its parts.

00354

2.6. Working with documents

You can choose from a number of possibilities when you want to open, save, or close documents. As a rule, these documents will be images. In addition, your software supports some other document types. You will find a list of supported documents in the online help.

Saving documents

You should always save important documents immediately following their acquisition. You can recognize documents that have not been saved by the star icon after the document's name.

There are a number of ways in which you can save documents.

1. To save a single document, activate the document in the document group and use the *File > Save As...* command.
2. Use the *Documents* tool window. Select the desired document and use the *Save* command in the context menu. For the selection of documents, the standard MS-Windows conventions for multiple selection are valid.
3. Use the *Gallery* tool window. Select the desired document and use the *Save* command in the context menu. For the selection of documents, the standard MS-Windows conventions for multiple selection are valid.
4. Save your documents in a database. That enables you to store all manner of data that belongs together in one location. Search and filter functions make it quick and easy to locate saved documents. Detailed information on inserting documents into a database can be found in the online help.

Autosave and close

1. When you exit your software, all data that has not yet been saved will be listed in the *Unsaved Documents* dialog box. This gives you the chance to decide which document you still want to save.
2. With some acquisition processes, the acquired images will be automatically saved after the acquisition has finished.
3. You can also configure your software in such a way that all images are saved automatically after image acquisition. To do so, use the *Acquisition Settings > Saving* dialog box. Here, you can also configure your software in such a way that all images are automatically saved in a database after the image acquisition.

Closing documents

There are a number of ways in which you can close documents.

1. Use the *Documents* tool window.
Select the desired document and use the *Close* command in the context menu. For the selection of documents, the standard MS-Windows conventions for multiple selection are valid.
2. To close a single document, activate the document in the document group and use the *File > Close* command. Alternatively, you can click the button with the cross [x]. You can find this button at the top right of the document tab located next to the document name.
3. Use the *Gallery* tool window.
Select the desired document and use the *Close* command in the context menu. For the selection of documents, the standard MS-Windows conventions for multiple selection are valid.

Closing all documents To close all loaded documents use the *Close All* command. You will find this command in the *File* menu, and in both the *Documents* and the *Gallery* tool window's context menu.

Closing a document immediately To close a document immediately without a query, close it with the [Shift] key depressed. Data you have not saved will be lost.

Opening documents

There are a number of ways in which you can open or load documents.

1. Use the *File > Open...* command.
2. Use the *File Explorer* tool window.
To load a single image, double click on the image file in the *File Explorer* tool window.
To load several images simultaneously, select the images and with the left mouse button depressed, drag them into the document group. For the selection of images the standard MS-Windows conventions for multiple selection are valid.
3. Drag the document you want, directly out of the MS-Windows Explorer, onto your software's document group.
4. To load documents from a database into the document group, use the *Database > Load Documents...* command. You can find more information in the online help.

Note: At the same time, up to 150 documents can be loaded in the document group.

Generating a test image If you want to get used to your software, then sometimes any image suffices to try out a function.
Press [Ctrl + Shift + Alt + T] to generate a color test image.
With the [Ctrl + Alt + T] shortcut, you can generate a test image that is made up of 256 gray values.

Activating documents in the document group

There are several ways to activate one of the documents that has been loaded into the document group and thus display it on your monitor.

1. Use the *Documents* tool window. Click the desired document there.
2. Use the *Gallery* tool window. Click the desired document there.
3. Click the title of the desired document in the document group.
4. To open a list with all currently loaded documents, use the [Ctrl + Tab] shortcut. Left click the document that you want to have displayed on your monitor.
5. Use the keyboard shortcut [Ctrl + F6] or [Ctrl + Shift + F6], to have the next document in the document group displayed. With this keyboard shortcut you can display all of the loaded documents one after the other.
6. In the *Window* menu, you will find a list of all of the documents that have been loaded. Select the document you want from this list.

Attaching a document to an e-mail

1. Load the documents you want to attach to your e-mail.
2. Use the *File > Send E-mail...* command.
3. Check whether all documents you want to attach are selected.
4. Click the *Send* button to generate an e-mail with the selected documents included as attachments.
 - You will receive a warning message if the sum of file sizes of all documents exceeds the maximum permitted size.
 - A new e-mail form will be opened by your e-mail program. Your e-mail program does not have to be already running for this to happen. The e-mail contains all of the selected image and document files as attachments.

As long as the e-mail form remains open, you cannot use your software or your e-mail program. The e-mail form cannot be minimized, no can other e-mails be generated, nor can you read any incoming e-mails. You can't close the *Send E-mail* dialog box nor continue working.
5. Enter the recipient's address and your message and then send off your e-mail.

00143

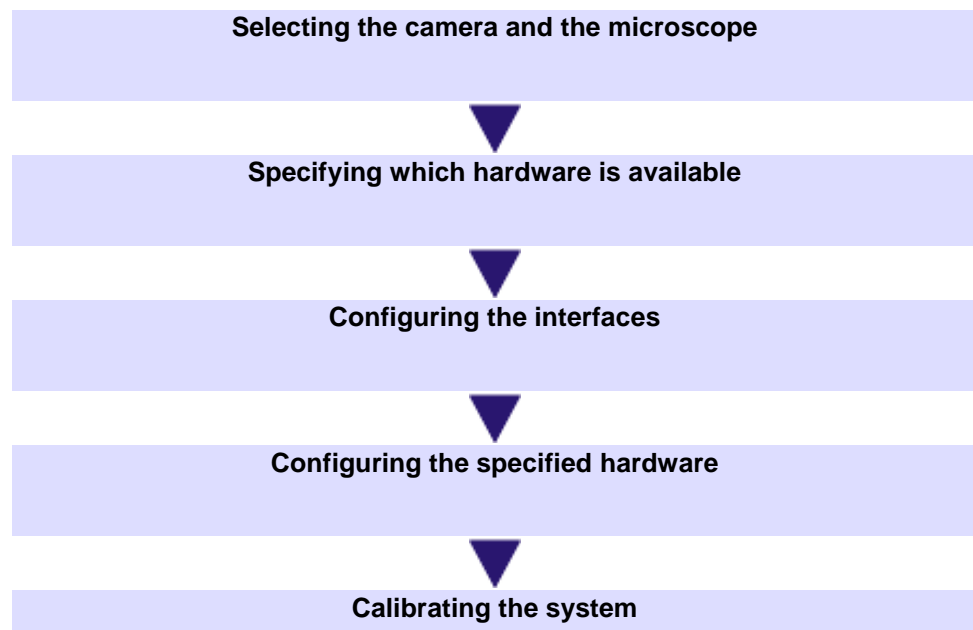
3. Configuring the system

Why do you have to configure the system?

After successfully installing your software you will need to first configure your image analysis system, then calibrate it. Only then will you be ready to acquire high quality and well calibrated images. When you work with a motorized microscope, you will also need to configure the existing hardware, to enable the program to control the motorized parts of your microscope.

Process flow of the configuration

To set up your system, the following steps are necessary:



Selecting the camera and the microscope

The first time you start your software after the installation has been made, a quick configuration with some default settings will be made. In this step you need only to specify the camera and microscope types, in the [Quick Device Setup](#) dialog box. The microscope will be configured with a selection of typical hardware components.

Specifying which hardware is available

Your software has to know which hardware components your microscope is equipped with. Only these hardware components can be configured and subsequently controlled by the software. In the [Acquire > Devices > Device List](#) dialog box, you select the hardware components that are available on your microscope.

If you use a preset configuration that was offered in the [Quick Device Setup](#) dialog box, check now whether your system is really equipped with the hardware components that are defined in the configuration.

Configuring the interfaces

Use the [Acquire > Devices > Interfaces](#) command, to configure the interfaces between your microscope or other motorized components, and the PC on which your software runs.

Configuring the specified hardware

If you use a preset configuration that was offered in the [Quick Device Setup](#) dialog box, you can skip this step. Usually various different devices, such as a camera, a microscope and/or a stage, will belong to your system. Use the [Acquire > Devices > Device Settings...](#) dialog box to configure the connected devices so that they can be correctly actuated by your software.

Calibrating the system When all of the hardware components have been registered with your software and have been configured, the functioning of the system is already ensured. However, it's only really easy to work with the system and to acquire top quality images, when you have calibrated your software. The detailed information that helps you to make optimal acquisitions, will then be available. Your software offers a wizard that will help you while you go through the individual calibration processes. Use the *Acquire > Calibrations...* command to start the software wizard.

About the system configuration

When do you have to configure the system? You will only need to completely configure and calibrate your system anew when you have installed the software on your PC for the first time, and then start it. When you later change the way your microscope is equipped, you will only need to change the configuration of certain hardware components, and possibly also recalibrate them.

Switching off your operating system's hibernation mode When you use the MS-Windows Vista operating system: Switch the hibernation mode off.

1. To do so, click the Start button located at the bottom left of the operating system's task bar.
2. Use the *Control Panel* command.
3. Open the *System and Maintenance > Power Options > Change when the computer sleeps* window.
 - Here, you can switch off your PC's hibernation mode.

When you use the **MS-Windows 7** operating system: Switch off your PC's power saving options and make sure that your PC does not automatically goes to hibernation mode.

1. To do so, click the Start button located at the bottom left of the operating system's task bar.
2. Click *Control Panel, System and Security* and then click *Power Options*.
3. On the *Select a power plan* page, click *Change plan settings*.
4. On the *Change settings for the plan* page, click *Change advanced power settings*.
 - Here, you can switch off your PC's power saving and hibernation mode.

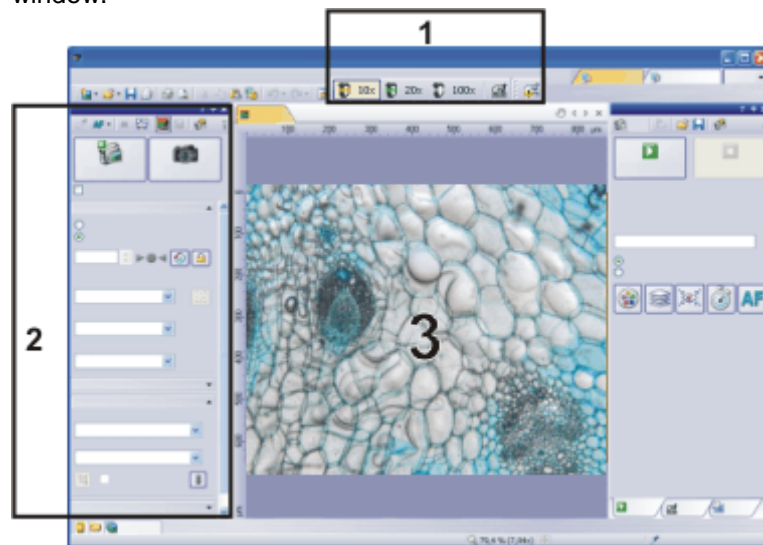
00159

4. Acquiring a single image

4.1. Snapshot

You can use your software to acquire high resolution images in a very short period of time. For your first acquisition you should carry out these instructions step for step. Then, when you later make other acquisitions, you will notice that for similar types of sample many of the settings you made for the first acquisition can be adopted without change.

1. Switch to the *Acquisition* layout. To do this, use, e.g., the *View > Layout > Acquisition* command.
 - You can find the *Microscope Control* (1) toolbar at the upper edge of the user interface, right below the menu bar. To the left of the document group, you see the *Camera Control* (2) tool window.



Selecting an objective

2. On the *Microscope Control* toolbar, click the button with the objective that you use for the image acquisition.

Switching on the live-image



Setting the image quality

3. In the *Camera Control* tool window, click the *Live* button.
 - The live-image (3) will now be shown in the document group.

4. Go to the required specimen position in the live-image.

5. Bring the sample into focus. The *Focus Indicator* toolbar is there for you to use when you are focusing on your sample.

6. Check the color reproduction. If necessary, carry out a white balance.

7. Check the exposure time. You can either use the automatic exposure time function, or enter the exposure time manually. Select the resolution you want.

Acquiring and saving an image



8. In the *Camera Control* tool window, click the *Snap* button.
 - The acquired image will be shown in the document group.

9. Use the *File > Save As...* command to save the image. Use the recommended TIF file format.

00027 05022014

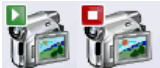
4.2. Behavior of the live window

The behavior of the live window depends on the acquisition settings in the *Acquisition Settings > Acquisition > General* dialog box.

Prerequisite

For the following step-by-step instructions, the *Keep document when live is stopped* option is selected and the *Create new document when live is started* check box is cleared.

Switching the live-image on and off without acquiring an image



1. Make the *Camera Control* tool window appear. To do this, use, e.g., the *View > Tool Windows > Camera Control* command.
2. In the *Camera Control* tool window, click the *Live* button.
 - A temporary live window named "Live (active)" will be created in the document group.
 - The live-image will be shown in the live window.
 - You can always recognize the live modus by the changed look of the *Live* button in the *Camera Control* tool window.
3. Click the *Live* button again.
 - The live mode will be switched off.
 - The active live image will be stopped.
 - The live window's header will change to "Live (stopped)". You can save the stopped live-image located in the live window just as you can every other image.

The live window may look similar to an image window, but it will be handled differently. The next time you switch on the live mode, the image will be overwritten. Additionally, it will be closed without a warning message when your software is closed.

Switching to the live-image and acquiring an image



1. Make the *Camera Control* tool window appear. To do this, use, e.g., the *View > Tool Windows > Camera Control* command.
2. In the *Camera Control* tool window, click the *Live* button.
 - A temporary live window named "Live (active)" will be created in the document group.
 - The live-image will be shown in the live window.
 - You can always recognize the live modus by the changed look of the *Live* button in the *Camera Control* tool window.
3. Click the *Snap* button.
 - The live mode will be switched off. The live window's header will change to "Live (stopped)".
 - At the same time, a new image document will be created and displayed in the document group. You can rename and save this image. If you have not already saved it when you end your software, you will be asked if you want to do so.

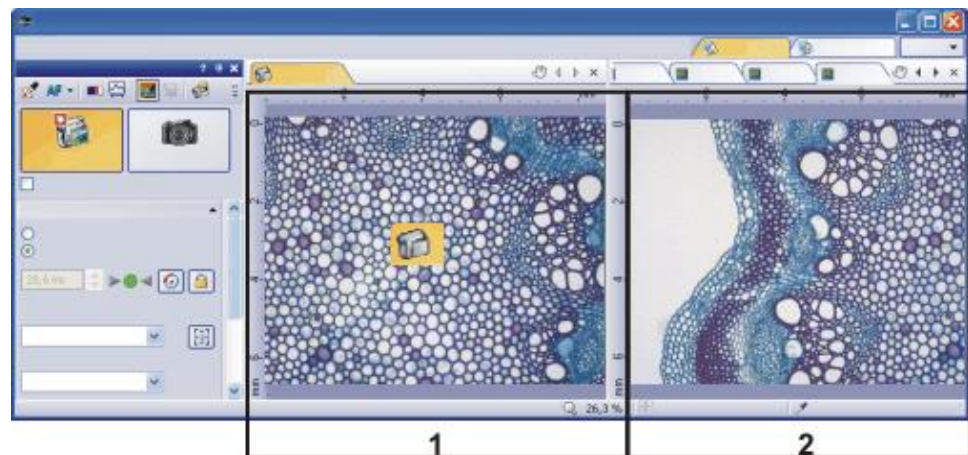
Displaying the live-image and the acquired images simultaneously

Task You want to view the live-image and the acquired images simultaneously. When you do this, it should also be possible to look through the acquired images without having to end the live mode.



1. Close all open documents.
2. Open the *Acquisition Settings > Acquisition > General* dialog box. To do so, click, e.g., the *Acquisition Settings* button on the *Camera Control* tool window.
3. There, make the following settings:
 - Choose the *Keep document when live is stopped* option.
 - Clear the *Create new document when live is started* check box.
 - Select the *Continue live after acquisition* check box.
4. Switch to the live mode. Acquire an image, then switch the live mode off again
 - Both of the image windows "Live (stopped)" and "Image_<Serial No.>" are now in the document group.
 - The "Live (stopped)" image window is active. That's to say, right now you see the stopped live-image in the document group. In the document bar, the Name "Live (stopped)" is highlighted in color.
5. Split the document group, to have two images displayed next to each other. That's only possible when at least two images have been loaded. That's why you created two images in the first step.
6. Use the *Window > Split/Unsplit > Split/Unsplit Document Group (Left)* command.
 - This command creates a new document group to the left of the current document group. In the newly set up document group the active document will be automatically displayed. Since in this case, the active document is the stopped live-image, you will now see the live window on the left and the acquired image on the right.
7. Start the live mode.
 - In the document group, the left window will become the live window Live (active)". Here, you see the live-image.
8. Activate the document group on the right. To do so, click, for example, the image displayed there.
9. Click the *Snap* button.
 - The acquired image will be displayed in the active document group. In this case, it's the document group on the right.
 - After the image acquisition has been made, the live-image will automatically start once more, so that you'll then see the live-image again on the left.
 - While the live-image is being shown on the left, you can switch as often as you want between the images that have up till then been acquired.





You can set up your software's user interface in such a way that you can view the live-image (1) and the images that have up till then been acquired (2), next to one another.

00181 25072011

4.3. Acquiring HDR images


HDR stands for **H**igh **D**ynamic **R**ange. Dynamic range relates to the capacity of cameras or software to display both bright and dark image segments. Before acquiring an HDR image, the necessary exposure range needs to be determined for the current sample. The exposure range is made up of a minimum and maximum exposure time as well as several exposure times between them. A recently determined exposure range will continue to be used for all HDR images until you let your software determine the exposure range anew. It is irrelevant whether the exposure range had been determined automatically or manually.

If you are acquiring several images of the same or similar parts of a sample, you don't need to determine the exposure range each time. If you change the sample or adjust settings on the microscope, it is recommended to determine the exposure range anew (either automatically or manually).

Acquiring an HDR image with a manually determined exposure range

With this procedure, you set the minimum and maximum exposure time in the *Camera Control* tool window yourself. Your software guides you through the process with relevant message boxes. How much the exposure time is adjusted by is determined by your software with regards to the minimum and maximum exposure time.

Preparations

1. Switch to the *Acquisition* layout. To do this, use, e.g., the *View > Layout > Acquisition* command.
2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the acquisition of the HDR image.
3. Switch to the live mode, and select the optimal settings for your acquisition, in the *Camera Control* tool window. Carry out a white balance. Choose an approximate exposure time.
4. Search for the part of the sample which you want to acquire an HDR image of. This should be a position which has such significant differences in brightness that not all segments can be shown with optimal lighting.
5. Finish the live mode.
6. In the *Camera Control* tool window, select the *Activate HDR* check box.
 - In the upper part of the tool window, the *Snap* button changes to the *HDR* button.
7. In the *Determine exposure range* group, click the *Manual...* button to define the exposure range for this acquisition anew.
 - The *Determine exposure range* message box appears. It prompts you to reduce the exposure time so far that enough image details can be recognized in the bright image segments and no segments are overexposed.
8. Change the exposure time in the *Exposure* group, which is part of the *Camera Control* tool window. Make sure that the *Manual* option is chosen. You can change the value by using the slide control or by entering an exposure time with the keyboard and pressing the [Enter] key. Check the live image on display. Once the bright image segments are no longer overexposed, click the *OK* button in the *Determine exposure range* message box.
 - By doing so, you have determined the lower limit of the exposure range (=the shortest exposure time).
9. Now, the *Determine exposure range* message box prompts you to raise the exposure time so high that the dark image segments are no longer underexposed. Change the exposure time in the *Exposure* group, which is part of the *Camera Control* tool window. Check the live image on display. Once the dark image segments are bright enough, click the *OK* button in the *Determine exposure range* message box.
 - By doing so, you have determined the upper limit of the exposure range (=the longest exposure time).
10. Click the *HDR* button in the *Camera Control* tool window to start the image acquisition.
 - The image acquisition will begin. Pay attention to the progress bar located in the status bar . It shows how long the acquisition has taken and the total acquisition time. The

Acquiring an HDR image



progress bar contains the *Cancel* button, which you can use to stop the current image acquisition.

- After the acquisition has been completed the HDR image will be shown in the document group.
11. Check the image. If you want to change the settings (to use a different algorithm for the output rendering, for example), open the *Acquisition Settings* dialog box. Select the *Acquisition > HDR* option in the tree view.
 12. If you don't want to change any settings, use the *File > Save As...* command to save the image. Use the recommended TIF or VSI file format.
 - These are the only formats which also save all the image information including the HDR entries together with the image. In this way, you can see, e.g., whether or not an image was acquired using HDR. Open the *Properties* tool window and look at the data in the *Camera* group.

Acquiring more HDR images without setting the exposure range anew

If you have just acquired HDR images of the same or a similar sample, as a rule, it is not necessary to determine the dynamic range anew. In this case, you have already completed the preparations for acquisition (such as carrying out a white balance) and set the HDR image acquisition settings correctly (such as choosing the optimal algorithm used for output rendering) anyway.

In such circumstances, acquiring an HDR image is especially easy. Do the following:

1. In the *Camera Control* tool window, select the *Activate HDR* check box.
2. Click the *HDR* button in the *Camera Control* tool window to start the image acquisition.
 - The image acquisition will begin. After the acquisition has been completed the HDR image will be shown in the document group.
3. Check the image before saving it.
 - This step can be left out if your software is configured to import images into a database directly after acquisition.


07500


5. Acquiring multi-dimensional images

What is a multi-dimensional image?

You can combine a series of individual images into one image. You can, e.g., assemble separate images that belong to different color channels. Depending on how the frames differ, the multi-dimensional image that results from their combination will also vary.

A standard image is two dimensional. The position of every pixel will be determined by its X- and Y-values. Fluorescence color, time and Z-position of the microscope stage are the possible additional dimensions of a multi-dimensional image.

 A multi-channel image as a rule, shows a sample that has been marked with several different fluorochromes. The multi-channel image is made up of a combination of the individual fluorescence images.

 In a time stack all frames have been acquired at different points of time. A time stack shows you how an area of a sample changes with time. You can play back a time stack just as you do a movie.


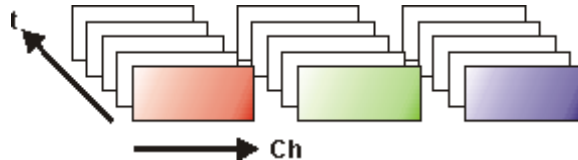
 A Z-stack contains frames acquired at different focus positions. You need a Z-stack, for instance, for the calculation of an EFI image.

Image containing several dimensions

The different multi-dimensional images can be arbitrarily combined. A multi-channel time stack, for instance, incorporates several color channels. Every color channel incorporated in the image is reproduced with its own time stack.



Navigation bar in the image window

The multi-dimensional images have their own navigation bar directly in the image window. Use this navigation bar to define how a multi-dimensional image is to be displayed in the image window.

00009

5.1. Overview - Acquisition processes

Your software offers a wide range of different acquisition processes.

Note: Your software is available in a variety of versions. This online help contains the description of all acquisition processes. For this reason, it can happen that your software version doesn't have some of the acquisition processes described here.

Basic acquisition processes



Complex acquisition processes



Basic acquisition processes

Use the *Camera Control* tool window to acquire images and movies.

Acquisition process -
Snapshot



You can use your software to acquire high resolution images in a very short period of time.

Acquisition process -
Movie



You can use your software to record a movie. When you do this, your camera will acquire as many images as it can within an arbitrary period of time. The movie will be saved as a file in the AVI format. You can use your software to play it back.

Complex acquisition processes

Use the *Process Manager* tool window to handle complex acquisition processes.

Acquisition process -
Time Lapse



With the automatic acquisition process *Time Lapse* you acquire a series of frames one after the other. This series of individual images makes up a time stack. A time stack shows you how an area of a sample changes with time. You can play back a time stack just as you do a movie.

You can combine the *Time Lapse* acquisition process with other acquisition processes. If your software supports the *Multi Channel* acquisition process, use, e.g., the *Time Lapse* acquisition process to acquire a multi-channel time stack.

AF When your microscope stage is equipped with a motorized Z-drive, use the *Z-Stack* acquisition process to acquire an EFI image. You'll find a description of the individual settings along with the description of the acquisition process.

Acquisition process - Z-
Stack



Use the automatic acquisition process *Z-Stack* to acquire a Z-stack. A Z-stack contains frames acquired at different focus positions. That is to say, the microscope stage was located in a different Z-position for the acquisition of each frame.

Alternatively, you can also acquire an EFI image with the *Z-Stack* acquisition process. Then a resulting image (EFI image) with a practically unlimited depth of focus is automatically calculated from the Z-stack that has been acquired. Such an image is focused throughout all of its segments. EFI is the abbreviation for Extended Focal Imaging.

You can combine the *Z-Stack* acquisition process with other acquisition processes.

If your software supports the *Multi Channel* acquisition process, you can combine the *Z-Stack* acquisition process with the multi channel acquisition to acquire a multi-channel Z-stack.

Acquisition process -
XY-Positions / MIA



You can only use this acquisition process when your microscope is equipped with a motorized XY-stage. With this acquisition process you can carry out one or more automatic acquisition processes at different positions on the sample or acquire a stitched image of a larger sample position.

AF If your microscope is equipped with a motorized Z-drive, you can switch on a software autofocus. You'll find a description of the individual settings along with the description of the acquisition process.

Acquisition process -
Multi Channel



With the automatic acquisition process *Multi Channel* you acquire a multi-channel fluorescence image.

You can combine, e.g., the *Multi Channel* acquisition process with the *Z-stack* acquisition process to acquire a multi-channel Z-stack.

Note: When you use the DP80 camera, please note the following restriction. When you acquire a transmitted light image simultaneously with a multi-channel fluorescence image, the *Multi Channel* acquisition process can't be combined with another acquisition process, for example the *Z-stack* acquisition process. This restriction protects the camera from being damaged by permanently toggling between the two CCDs the camera provides.

Acquisition process -
Instant EFI



Use the manual acquisition process *Instant EFI* to acquire an EFI image at the camera's current position, that is sharply focused all over.

You can find more information on this acquisition process in the online help.

Acquisition process -
Manual MIA



When you use the *Manual MIA* acquisition process, you move the stage manually in such a way that different, adjoining sample areas are shown. With this acquisition process, you combine all of the images that are acquired, directly during the acquisition, just like a puzzle, into a stitched image. The stitched image will display a large sample segment in a higher X/Y-resolution than would be possible with a single acquisition.

Combination of several acquisition processes

You can combine several automatic acquisition processes. To do so, click the corresponding button for each acquisition process you require.

Note: Which automatic acquisition processes you can combine with each other, depends on your software.

The order of the automatic acquisition processes in the *Process Manager* tool window (from left to right) corresponds to the order in which the acquisition processes were carried out.

Examples



If you combine the two acquisition processes *Multi Channel* and *Z-Stack*, a complete multi-channel image will be acquired at every focus position. The *Multi Channel* acquisition process will then be carried out first. Only after this has been done, will the stage's Z-position be changed, and another multi-channel image acquired.



When you combine the two acquisition processes *Z-Stack* and *XY-Positions / MIA* to acquire a Z-stack at several positions on your sample, to begin with, the complete Z-stack at the first position will be acquired. When that has been done, your system will move to the next position, and will acquire the next Z-stack etc..

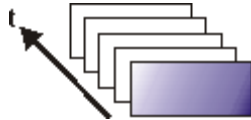
00442 13012011

6. Acquiring image series

With your software you can acquire time stacks, movies, and Z-stacks.

6.1. Time stacks

What is a time stack?




You can combine a series of individual images into one image. In a time stack all frames have been acquired at different points of time. A time stack shows you how an area of a sample changes with time. You can play back a time stack just as you do a movie.

A standard image is two dimensional. The position of every pixel will be determined by its X- and Y-values. With a time stack, the time when the image was acquired is an additional piece of information or "dimension" for each frame.

The frames making up a time stack can be 8-bit gray-value images, 16-bit gray-value images or 24-bit true-color images.

Note: A time stack can also be an AVI video. You can load and play back the AVI file format with your software.

How do I recognize a time stack?

You can immediately recognize the different image types by the icon which appears in front of the image name in the document group, or in the *Documents* tool window. When it is a time stack, this icon will be supplemented by a small clock. A time stack that is made up of true-color images has, e.g., this icon .

In the *Properties* tool window, you can use the *Frame Count* entry to find out how many individual images are contained in any given image.

Creating time stacks

There are different ways in which you can generate a time stack.

To acquire a time stack, use one of the two acquisition processes *Time Lapse* or *Movie*.

Use the *Image > Combine Frames...* command, to have several individual images combined into a time stack. A description of the *Combine Frames* dialog box can be found in the online help.

Displaying time stacks

A time stack contains much more data than can be displayed on your monitor.

A time stack will automatically have its own navigation bar directly in the image window. Use this navigation bar to determine which of the frames from a time stack is to be displayed on your monitor. You can also play back a time stack just as you would a movie.

You can find more information on the navigation bar in the online help.

Alternatively, you can also use the *Dimension Selector* tool window to determine how a time stack is to be displayed on your monitor, or to change this.

Hiding the navigation bar

You can also hide the navigation bar. To do this, use the *Tools > Options...* command. Select the *Images > General* entry in the tree view. Clear the *Show image navigation toolbar* check box.

Saving time stacks

When you save time stacks, you will, as a rule, use the VSI file format. Only when you use this file format is there no limit to the size a time stack can be. When you save smaller time stacks, you can also use the TIF or the AVI file format. With any other file format you will lose most of the image information during saving.

Use the *File > Save As...* command.

Converting time stacks Use the *Image > Separate > Time Frames* menu command to have a time stack broken down into selected individual images.

It is possible that, within a time stack, only a short period of time interests you. Use the *Extract* command to create a new time stack that only contains a selection of frames, from an existing time stack. In this way, you will reduce the number of frames within a time stack to only those that interest you. You will find this command in the context menu in the tile view for time stacks.

When you save a time stack in another file format as TIF or VSI, the time stack will also be converted. The time stack will then be turned into a standard true-color image. This image shows the frame that is at that moment displayed on the monitor.

Processing time stacks Image processing functions, e.g., a sharpen filter, affect either the whole image or only a selection of individual images. You will find most of the image processing operations in the *Process* menu. The dialog box that is opened when you use an image processing operation is made up in the same way for every operation. In this dialog box, select the *Apply on > Selected frames and channels* option to determine that the function only affects the selected frames.

Select the *Apply on > All frames and channels* option, to process all of the individual images.

Select the individual images that you want to process, in the tile view. You can find more information on this image window view in the online help. Look through the thumbnails and select the images you want to process. In the tile view, the standard MS-Windows conventions for multiple selection are valid.

An image processing operation does not change the source image's dimensions. The resulting image is, therefore, comprised of the same number of separate images as the source image.

00011

Time Lapse / Movie

Both the *Time Lapse* and the *Movie* acquisition processes document the way a sample changes with time. What is the difference between the two processes?

When is it better for me to acquire a time stack?

Use the *Time Lapse* acquisition process in the following cases:

- Use the *Time Lapse* acquisition process when processes that run slowly are to be documented, e.g., where an acquisition is to be made only every 15 minutes.
- Use the "Time Lapse" acquisition process when, while the acquisition is in progress, you want to see the frames that have already been acquired, for example, to check on how an experiment is progressing. To do this, click the *Tile View* button in the navigation bar in the image window.
- Use the "Time Lapse" acquisition process when you want to use those of your software's additional functions that can only be saved in the VSI or TIF file format. For example, to measure objects, to insert drawing elements such as arrows, or a text, or to have the acquisition parameters for the camera and microscope that you've used, available at any time in the future.
- Use the "Time Lapse" acquisition process when the important thing is to achieve an optimal image quality, and the size of the file is no problem.

Saving a time stack as an AVI

You can also save a time stack as an AVI file, at a later date. To do this, load the time stack into the document group, select the *File > Save as...* command, and select the AVI file type. Make, if necessary, additional settings in the *Select AVI Save Options* dialog box.

When is it better for me to acquire a movie?

Use the "Movie" acquisition process in the following cases:

- Use the "Movie" acquisition process when processes that run very quickly are to be documented (the number of acquisitions per second is considerably higher with movies than with time stacks).
- Use the "Movie" acquisition process when you want to give the movie to third persons who do not have this software (AVI files can also be played back with the MS Media Player).
- Use the "Movie" acquisition process when keeping file sizes small is of great importance.

00107

Acquiring a movie

You can use your software to record a movie. When you do this, your camera will acquire as many images as it can within an arbitrary period of time. The movie will be saved as a file in the AVI format. You can use your software to play it back.

Selecting an objective

Selecting the storage location



Selecting the compression method

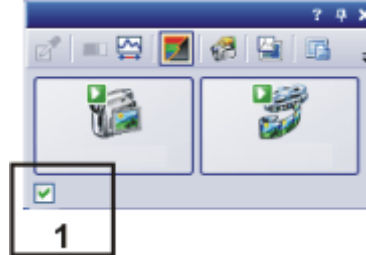
Setting the image quality

1. Switch to the "Acquisition" layout. To do this, use, e.g., the *View > Layout > Acquisition* command.
2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the movie acquisition.
3. In the *Camera Control* tool window's toolbar, click the *Acquisition Settings* button.
 - The *Acquisition Settings* dialog box opens.
4. Select the *Saving > Movie* entry in the tree structure.
5. You have to decide how a movie is to be saved after the acquisition. Select the *File system* entry in the *Automatic save > Destination* list to automatically save the movies you have acquired.
 - The *Path* field located in the *Directory* group shows the directory that will currently be used when your movies are automatically saved.
6. Click the [...] button next to the *Path* field to alter the directory.
 - The AVI file format is preset in the *File type* list. This is a fixed setting that cannot be changed.
7. Click the *Options...* button when you want to compress the AVI file in order to reduce the movie's file size.
8. From the *Compression* list, select the *M-JPEG* entry and confirm with *OK*.

Please note: Compressing the movie is only possible if the selected compression method (codec) has already been installed on your PC. If the compression method has not been installed the AVI file will be saved uncompressed.
The selected compression method must also be available on the PC that is used for playing back the AVI. Otherwise the quality of the AVI may be considerably worse when the AVI is played back.
9. Close the *Acquisition Settings* dialog box with *OK*.
10. Switch to the live mode, and select the optimal settings for movie recording in the *Camera Control* tool window. Pay special attention to setting the correct exposure time.
 - This exposure time will not be changed during the movie recording.
11. Find the segment of the sample that interests you and focus on it.

Switching to the "Movie recording" mode

12. Select the *Movie recording* check box (1). The check box can be found below the *Live* button in the *Camera Control* tool window.



- The *Snap* button will be replaced by the *Movie* button.


Starting movie recording



13. Click the *Movie* button to start the movie recording.

- The live-image will be shown and the recording of the movie will start immediately.
- In the status bar a progress indicator is displayed. At the left of the slash the number of already acquired images will be indicated. At the right of the slash an estimation of the maximum possible number of images will be shown. This number depends on your camera's image size and cannot exceed 2GB.



- This icon  on the *Movie* button will indicate that a movie is being recorded at the moment.

Stopping movie recording



14. Click the *Movie* button again to end the movie recording.

- The first image of the movie will be displayed.
- The navigation bar for time stacks will be shown in the document group. Use this navigation bar to play the movie.
- The software will remain in the "Movie recording" mode until you clear the *Movie recording* check box once more.

Acquiring a time stack

In a time stack all frames have been acquired at different points of time. With a time stack you can document the way the position on the sample changes with time. To begin with, for the acquisition of a time stack make the same settings in the *Camera Control* tool window as you do for the acquisition of a snapshot. Additionally, in the *Process Manager* tool window, you have to define the time sequence in which the images are to be acquired.

Task You want to acquire a time stack over a period of 10 seconds. One image is to be acquired every second.

Selecting an objective

Setting the image quality

Selecting the acquisition process



1. Switch to the "Acquisition" layout. To do this, use, e.g., the *View > Layout > Acquisition* command.
2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the image acquisition.
3. Switch to the live mode, and select the optimal settings for your acquisition, in the *Camera Control* tool window. Pay special attention to setting the correct exposure time. This exposure time will be used for all of the frames in the time stack.
4. Choose the resolution you want for the time stack's frames, from the *Resolution > Snap/Process* list.

5. Find the segment of the sample that interests you and focus on it.

6. Activate the *Process Manager* tool window.

7. Select the *Automatic Processes* option.

8. Click the *Time Lapse* button.

- The button will appear clicked. You can recognize this status by the button's colored background.
 - The [t] group will be automatically displayed in the tool window.
9. Should another acquisition process be active, e.g., *Z-stack*, click the button to switch off the acquisition process.


• The group with the various acquisition processes should now look like this:



Selecting the acquisition parameters

10. Clear the check boxes *Start delay* and *As fast as possible*.

11. Specify the time that the complete acquisition is to take, e.g., 10 seconds. Enter the value "00000:00:10" (for 10 seconds) in the *Recording time* field. You can directly edit every number in the field. To do so, simply click in front of the number you want to edit.

12. Select the radio button on the right-hand side of the field to specify that the acquisition time is no longer to be changed. The  lock icon will automatically appear beside the selected radio button.

13. Specify how many frames are to be acquired. Enter e.g., 10 in the *Cycles* field.

• The *Interval* field will be updated. It shows you the time that will elapse between two consecutive frames.

Acquiring a time stack



14. Click the *Start* button.

• The acquisition of the time stack will start immediately.

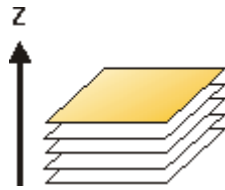


- The *Start Process* button changes into the *Pause* button. A click on this button will interrupt the acquisition process.
- The *Stop* button will become active. A click on this button will stop the acquisition process. The images of the time stack acquired until this moment will be preserved.
- At the bottom left, in the status bar, the progress bar will appear. It informs you about the number of images that are still to be acquired.
- The acquisition has been completed when you can once more see the *Start* button in the *Process Manager* tool window, and the progress bar has been faded out.
- You will see the time stack you've acquired in the image window. Use the navigation bar located in the image window to view the time stack. You can find more information on the navigation bar in the online help.
- The time stack that has been acquired will be automatically saved. The storage directory is shown in the *Acquisition Settings > Saving > Process Manager* dialog box. The preset file format is VSI.

Note: When other programs are running on your PC, for instance a virus scanning program, it can interfere with the performance when a time stack is being acquired.

00304 12012011


6.2. Z-stack



What is a Z-stack? You can combine a series of separate images into one image file. A Z-stack contains frames acquired at different focus positions. A Z-stack is needed, e.g., for calculating an EFI image by the *Process > Enhancement > EFI Processing...* command.

A standard image is two dimensional. The position of every pixel will be determined by its X- and Y-values. With a Z-stack, the focus position or the height of the sample is an additional item of information for every pixel.

The frames making up a Z-stack can be 8-bit gray-value images, 16-bit-gray-value images or 24-bit-true-color images.

How do I recognize a Z-stack? You can immediately recognize a multi-dimensional image by its icon which appears in front of the image name in the document group or in the *Documents* tool window. When it is a Z-stack, this icon will be supplemented by a small Z .

Creating a Z-stack There are different ways in which you can generate a Z-stack.

- To acquire a Z-stack, use the "Z-Stack" acquisition process.
- Use the *Image > Combine Frames...* command to have several separate images combined into a Z-stack.

Displaying a Z-stack A Z-stack contains much more data than can be displayed on your monitor. A Z-stack image will automatically have its own navigation bar directly in the image window. Use this navigation bar to determine which of the frames from a Z-stack is to be displayed on your monitor. You can also play back the Z-stack just as you would a movie. A detailed description of the navigation bar can be found in the online help.



Alternatively, you can also use the *Dimension Selector* tool window to define how a Z-stack image is to be displayed on your monitor, or to change this.

Saving a Z-stack Please note: Z-stacks can only be saved in the TIF or VSI file format. Otherwise they lose a great deal of their image information during saving.

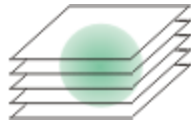
00012

Acquiring a Z-stack

A Z-stack contains frames acquired at different focus positions. That is to say, the microscope stage was located in a different Z-position for the acquisition of each frame.

Note: You can only use the *Z-Stack* acquisition process when your stage is equipped with a motorized Z-drive.

Example: You want to acquire a Z-stack. The sample is approximately 50 μm thick. The Z-distance between two frames is to be 2 μm .



Selecting an objective

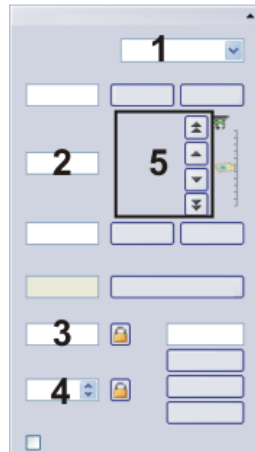
Setting the image quality

Selecting the acquisition process

1. Switch to the *Acquisition* layout. To do this, use, e.g., the *View > Layout > Acquisition* command.
2. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the image acquisition.
3. Switch to the live mode, and select the optimal settings for your acquisition, in the *Camera Control* tool window. Pay special attention to setting the correct exposure time. This exposure time will be used for all of the frames in the Z-stack.
4. Search out the required position in the sample.
5. Activate the *Process Manager* tool window.
6. Select the *Automatic Processes* option.
7. Click the *Z-Stack* button.
 - The button will appear clicked. You can recognize this status by the button's colored background.
 - The [Z] group will be automatically displayed in the tool window.
8. Should another acquisition process be active, e.g., *Multi Channel*, click the button to switch off the acquisition process.
 - The group with the various acquisition processes should now look like this:



Selecting the acquisition parameters



Set the acquisition parameters for the acquisition of a Z-stack in the *Process Manager* tool window. Use the fields and buttons (1-4) for this.

9. Select the *Range* entry in the *Define* list (1).
10. Enter the Z-range you want, in the *Range* field (2). In this example, enter a little more than the sample's thickness (= 50 μm), e.g., the value 60.
11. In the *Step Size* field (3), enter the required Z-distance, e.g., the value 2, for a Z-distance of 2 μm . The value should roughly correspond to your objective's depth of focus.
 - In the *Z-Slices* field (4) you will then be shown how many frames are to be acquired. In this example 31 frames will be acquired.
12. Find the segment of the sample that interests you and focus on it. To do this, use the arrow buttons (5). The buttons with a double arrow move the stage in larger steps.
13. Click the *Start* button.

Acquiring an image



- Your software now moves the Z-drive of the microscope stage to the start position. The starting position lies half of the Z-range deeper than the stage's current Z-position.
- The acquisition of the Z-stack will begin as soon as the starting position has been reached. The microscope stage moves upwards step by step and acquires an image at each new Z-position.
- The acquisition has been completed when you can once more see the *Start* button in the *Process Manager* tool window, and the progress bar has been faded out.
- You can see the acquired Z-stack in the image window. Use the navigation bar located in the image window to view the Z-stack. You can find more information on the navigation bar in the online help.
- The Z-stack that has been acquired will be automatically saved. You can view the storage directory in the *Acquisition Settings > Saving > Process/Experiment* dialog box. The preset file format is VSI.

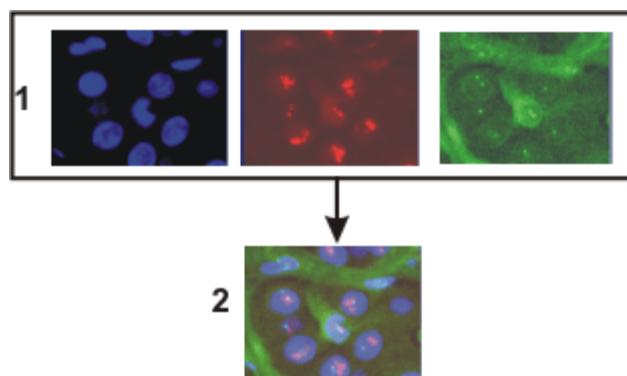
Note: When other programs are running in the background on your PC, for instance a virus scanning program, it can interfere with the performance when a Z-stack is being acquired.

00367 10072013

7. Acquiring fluorescence images

What is a multi-channel image?

A multi-channel image combines a series of monochrome images into one image. The multi-channel image usually shows a sample that has been stained with several different fluorochromes. The multi-channel image is made up of a combination of the individual fluorescence images. You can have the individual fluorescence images displayed separately or also as a superimposition of all of the fluorescence images.



At the top of the illustration you can see the individual fluorescence images (1). Below, you can see the superimposition (2) of the separate fluorescence images.


The separate images' image type

The separate images making up a multi-channel image can be 8-bit gray-value images or 16-bit gray-value images.

Combination with a times stack or Z-stack

A multi-channel image can be combined with a time stack or a Z-stack. A multi-channel time stack, for instance, then incorporates several color channels. Every color channel incorporated in the image is reproduced with its own time stack.

How do I recognize a multi-channel image?

You can immediately recognize a multi-channel image by this icon  which appears in front of the image name in the document group or in the *Documents* tool window.

In the *Properties* tool window, you can use the *Channels* entry to find out how many channels are contained in any given image.

Creating multi-channel images

Your software offers you several ways of acquiring a multi-channel image.

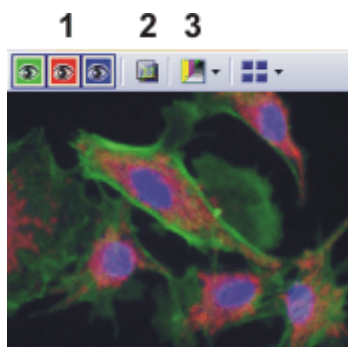
Use the *Multi Channel* automatic acquisition process to acquire a multi-channel image.

Use the *Image > Combine Channels...* command to have several separate images combined into a multi-channel image.

Displaying multi-channel images

A multi-channel image contains much more data than can be displayed on your monitor.

Using the navigation bar



When you load a multi-channel image into your software, you'll see a navigation bar in the image window, that provides you with access to all of the fluorescence channels.

- You can have the individual fluorescence images displayed separately or also as a superimposition of all of the fluorescence images (1).
- Should you have acquired a brightfield of the sample together with the fluorescence images, you can make this brightfield appear or disappear (2).
- The individual fluorescence images are monochrome. For this reason, you can change the color mapping however you like. You can display the fluorescence channels in the fluorescence colors, use a pseudo color table of your choice, or also display the original images (3).

Hiding the navigation bar

You can also hide the navigation bar. To do this, use the *Tools > Options...* command. Select the *Images > General* entry in the tree view. Clear the *Show image navigation toolbar* check box.

Using the "Dimension Selector" tool window

Alternatively, you can also use the *Dimension Selector* tool window to define how a multi-channel image is to be displayed on your monitor, or to change this. There you can, for example, change the fluorescence colors for individual color channels.

Saving multi-channel images

Please note: Multi-channel images can only be saved in the TIF or VSI file format. Otherwise they lose a great deal of their image information during saving.

Converting multi-channel images

Breaking down a multi-channel image into its color channels

Use the *Separate* command to have a multi-channel image broken down into chosen color channels. The resulting images are still of the "multi-channel" type, contain though, only one color channel.

There are several ways of accessing this command:

- Click the *Separate Channels* button in the *Dimension Selector* tool window.
- Use the *Separate* command from the *Dimension Selector* tool window's context menu.
- Use the *Image > Separate > Channels* menu command.

Reducing the color channels in a multi-channel image

Use the *Extract* command to create a new multi-channel image that is made up of fewer color channels than the source image.

Select all of the color channels you wish to retain, in the *Dimension Selector* tool window. Then use the *Extract* command in the tool window's context menu.

Converting multi-channel images while saving them

When you save a multi-channel image in another file format as TIF or VSI, the multi-channel image will also be converted. The multi-channel image then becomes a standard 24-bit true-color image. This image will always show exactly what is currently displayed on your monitor, that is to say, e.g., the superimposition of all of the channels or possibly only one channel.

Processing multi-channel images

Image processing functions, e.g., a sharpen filter, affect either the whole image or only a selection of individual images. You will find most of the image processing operations in the *Process* menu.

Select the frames that you want to process in the *Dimension Selector* tool window. The channels you have selected will be highlighted in color in the tool window.

The dialog box that is opened when you use an image processing operation is made up in the same way for every operation. In this dialog box, select the *Apply on > Selected frames and channels* option to determine that the function only affects the selected frames.

Select the *Apply on > All frames and channels* option, to process all of the individual images.

An image processing operation does not change the source image's dimensions. The resulting image is, therefore, comprised of the same number of separate images as the source image.

00010

7.1. Before and after you've acquired a fluorescence image

Before you acquire a fluorescence image

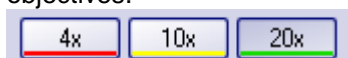
Defining observation methods

1. Define observation methods for your color channels.

Setting up the microscope for the acquisition of a fluorescence image

1. Use the *View > Tool Windows > Microscope Control* command to make the *Microscope Control* tool window appear.

- In the *Objectives* group, you can find the buttons you use to change objectives.



- In the *Observation method* group you can find a button for every observation method that has been defined. Observation methods should have been defined at least for brightfield and for every color channel.

2. Click the required objective's button.



3. Click the button for the observation method with the excitation that has the longest excitation wavelength (e.g., Red).

Setting up the camera for the acquisition of a fluorescence image

1. Use the *View > Tool Windows > Camera Control* command to make the *Camera Control* tool window appear.

2. Set the image resolution for the acquisition. With a high objective magnification, you require a lower resolution.

For this purpose, select the required resolution from the *Snap/Process* list, located in the *Resolution* group.

3. Reduce the image resolution in the live mode. When you use a higher binning in the live mode, the frame rate will be reduced, which enables you to focus better.

For this purpose, select an entry, e.g., with the supplement *Binning 2x2*, from the *Live/movie* list, located in the *Resolution* group.



4. Should you work with a color camera: Switch on your camera's grayscale mode. The *Toggle RGB/Grayscale mode* button should now look like this. You can find this button on the *Camera Control* tool window's toolbar.



5. If it's possible to set different bit depths with your camera, click the *Toggle Bitdepth* button to set the maximum bit depth.

Switching off the corrections for brightfield acquisitions



1. Use the *View > Toolbars > Calibrations* command to have the *Calibrations* toolbar displayed.

2. Switch off the white balance and the shading correction. To do that, release these buttons if they are there and available.

Focusing a fluorescence sample



1. Select the automatic exposure time.

2. In the *Camera Control* tool window, click the *Live* button.

- Should the live-image be too dark, select a higher value in the *Camera Control > Exposure > Exposure compensation* list.

- Should the exposure time become longer than 300 ms, reduce it by increasing the sensitivity or gain.

3. Bring the sample into focus.



- In the camera's black & white mode, you can reduce the diffused light. Click the *Online-Deblur* button, located in the *Camera Control* tool window's toolbar. You can then decide whether or not you want to apply the deconvolution filter. It's possible that you might have to lengthen the exposure time via the exposure time correction.

4. Finish the live mode. To do so, click the *Live* button in the *Camera Control* tool window.

Setting the storage location

Multi-channel images will be saved by default, as soon as the acquisition has been completed. As file format, the VSI file format will be used.

1. Before you start the acquisition, specify where the file is to be saved.

2. To do this, click the *Acquisition Settings* button, located in the *Process Manager* tool window's toolbar. Select the *Saving > Process Manager* entry in the tree view.

- You can find the current directory in the *Directory > Path* field.

3. Click the [...] button next to the *Path* field to change the directory into which the image is to be saved after its acquisition.

After you've acquired a fluorescence image

Viewing a multi-channel image

A multi-channel image is made up of the individual fluorescence images. You can set which color channels, resp. combination of color channels, will be displayed on your monitor. To do this, use the navigation bar in the image window.

Click a color field to make the channel appear or disappear. All of the color channels that are at the moment displayed on your monitor will be identified by an eye icon.

The navigation bar also offers you additional possibilities for changing the appearance of the multi-channel image.

You can find more information on the navigation bar in the online help.

"Properties" tool window

Numerous acquisition parameters will be saved together with the image.

Use the *View > Tool Windows > Properties* command to make the *Properties* tool window appear. In the *Properties* tool window, you can find that every color channel has its own *Channel* information group. This contains the channel name, the emission wavelength, the name of the observation method and the exposure time.

Saving multi-channel images

The multi-channel image will be automatically saved. You can set the storage directory in the *Acquisition Settings > Saving > Process Manager* dialog box. The file format used is VSI.

For the VSI format, a JPEG compression of 90% is preset. You can change the compression in the *Acquisition Settings* dialog box under *Saving > Process Manager > Automatic save > Options...*

00363

7.2. Defining observation methods for the fluorescence acquisition

Before you acquire a fluorescence image, you have to define observation methods for your color channels. Usually, observation methods that you can adapt for your microscope configuration, have already been predefined.

Prerequisite The system has already been configured and calibrated. For this purpose, you have to enter your hardware components in the *Acquire > Devices > Device List* dialog box, and configure them in the *Device Settings* dialog box. To finalize this action, calibrate your system by using the *Acquire > Calibrations...* command.

The tables that follow, contain example configurations for both motorized, and not motorized microscopes. Only the hardware components that are relevant for the acquisition of multi-channel fluorescence images are listed.

Device List		Example entries	
		Non-motorized microscope	Motorized microscope
Microscope			
Frame	<Name of your microscope>	BX51	BX61
Nosepiece	<Name of your nosepiece>	Manual Nosepiece	Motorized (UCB)
Mirror turret	<Name of your mirror turret>	Manual mirror turret	BX-RFAA
Stage			
Z-axis	<Name of your controller for the stage's Z-drive>	Not Motorized	Motorized (UCB)
Fluorescence/reflected light path			
Shutter	<Name of the reflected light shutter>	Manual Shutter	Motorized (UCB)
Transmission light path			
Lamp	<Name of your transmission lamp>	Not used	UCB Halogen-Lamp
Condenser	<Name of your condenser>	Manual Condenser	U-UCD8A


Device Settings	Entries
Nosepiece	<Your objectives>
Mirror turret	U-MNU U-MWB U-MWG For a position where there is no mirror cube, select the <i>Free</i> entry.
Condenser	With a motorized condenser: The hardware components <i>Aperture Stop</i> and <i>Top Lens</i> are additionally listed under the device settings.


Defining the observation method for transmission brightfield

Example: Hardware components in transmission brightfield: In transmission brightfield, there is no fluorescence mirror cube in the light path. The reflected light shutter is closed.


1. Use the *Acquire > Devices > Device Customization...* command. Activate the *Observation Methods* tab.
2. Click the *New Observation Method* button.
 - A dialog box, in which you can enter a name, will be opened.
3. Give the new observation method a name, then close the dialog box with *OK*. You could name your observation method, e.g., MyBF.
4. Select the mirror turret in the *Available components* list.
 - In the central area of the dialog box, the settings for the hardware components that have been selected, will be displayed.
5. In transmission brightfield, the mirror turret isn't allowed to contain a mirror cube. Therefore, select the *Adjust* entry in the *Status* list, located in the middle of the dialog box, and in the list that's below that one select the *Free* entry.
 - For manual microscopes: Where a manual microscope is concerned, the mirror turret can't be automatically moved to the position you want. For this reason, when you use a manual microscope, you'll receive a message that asks the user to make this setting manually. This message will also appear when you are defining the observation method. Confirm the message with *OK*.
6. Select the condenser.
 - Select the *Adjust* entry in the *Status* list, located in the middle of the dialog box, and in the list that's below that one, the *Free* entry.
7. Select the hardware component *Aperture Stop*.
 - Select the *Adjust* entry in the *Status* list, located in the middle of the dialog box. Enter "75%" in the field below that.
8. Select the hardware component *Top Lens*.
 - Select the *Adjust per objective* entry in the *Status* list.
 - In the middle of the *Device Customization* dialog box, you can now set the top lens for each objective separately.
 - The top lens is only used for objectives with higher magnifications (upwards of 10x) and is swung out for lower magnifications.



 *Setting up the mirror turret*

 *For motorized microscopes: Setting up the condenser*

9. Specify for which objectives the top lens should be brought into the light path and for which objectives it should be removed from the light path. To do so, select the *Use with this objective* check box for all objectives. In the *Selected components* list, each objective with a lower magnification than 10x needs to show the *Out* status. If that isn't the case, click in the middle of the dialog box on the *Out* button. In the *Selected components* list, each objective with a higher magnification than 10x or exactly 10x needs to show the *In* status. If that isn't the case, click in the middle of the dialog box on the *In* button.

 For motorized microscopes: Switching on the transmission lamp

10. Select the transmission lamp. You will find this lamp in the *Available components* list, under the *Transmission* entry.
11. Select the *Adjust* entry in the *Status* list, located in the middle of the dialog box. Set 9 V for the lamp. Use the button showing a small lamp to switch on the lamp.

- The button looks like this, if the lamp is switched on.



- In the *Selected components* list, you can also see that the lamp for this observation method will be switched on.





Saving the observation method


12. Click the *OK* button, to save the new observation method.
- The *Microscope Control* tool window will then contain a new button with this observation method's name.
 - You can now use the observation method in the *Process Manager* tool window, for the acquisition of a multi-channel fluorescence image.

Defining observation methods for fluorescence channels


Task Define an observation method for the acquisition of a fluorescence image. It also makes sense to do this when you don't work with a motorized microscope, since then the acquired image will be automatically colored with the correct fluorescence color.

Summary The following hardware components belong to an observation method for fluorescence channels. How you integrate these hardware components with the observation method, is described in detail in these step-by-step instructions.


Hardware component	Settings
 Fluorochromes	Assign the fluorescence colors.
 Mirror turret	Choose the mirror cube to be used.
 Fluorescence shutter	Open the fluorescence shutter for the image acquisition.
 Transmitted light lamp	Switch off the transmitted light lamp.

1. Use the *Acquire > Devices > Device Customization...* command. Activate the *Observation Methods* tab.
2. Click the *New Observation Method* button.
3. Give the new observation method a name, then close the dialog box with *OK*. Name the observation method, e.g., Blue.
4. Assign the fluorescence channel a fluorochrome (e.g., Blue or DAPI) and a color (e.g., Blue at 470 nm). To do so, select the *Fluorochromes*  entry in the *Available components* list. Select the *Use* entry in the *Status* list. In the *Fluorochrome* list located below that one, select the fluorochrome to be used, e.g., the entry *Blue* or *DAPI*. You can change the fluorescence color in the *Color* list, should that be necessary.




 Defining the fluorochrome

- It is important in all cases to define the fluorochrome for a fluorescence observation method, even if you don't automatically change any device settings. When the fluorescence color is linked to the observation method, every image you acquire with this observation method will be automatically colored in the corresponding color. This is valid irrespective of whether you work with a manual or a motorized microscope.


 For motorized microscopes: Setting up the mirror turret

It can make sense to use this setting and the additional settings for motorized microscopes also for manual microscopes. When you use a manual microscope, you'll receive a message that prompts you, to make this setting manually. As well as this, the device settings are saved together with the acquired image.

5. Select the mirror turret in the *Available components* list.
 - In the central area of the dialog box, the settings for the hardware components that have been selected, will be displayed.
6. For the acquisition of a fluorescence image a specific mirror cube has to be used. Therefore, select the *Adjust* entry in the *Status* list, located in the middle of the dialog box, and in the list that's below that one, select the mirror cube you want.

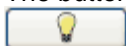
 For motorized microscopes: Setting the shutter for the fluorescence light path

7. In the *Available components* list, select the shutter that is located below the *Fluorescence/reflected* entry.
8. When the fluorescence acquisition is made, this shutter must be open. Therefore, select the *Use for acquisition* entry in the *Status* list, located in the middle of the dialog box.
 - Then the shutter will be opened before the image acquisition is made, and closed when this has been done, to avoid bleaching of the sample.


 For motorized microscopes: Switching off the transmission lamp

9. Select the transmission lamp. You will find this lamp in the *Available components* list, under the *Transmission* entry.
10. For the acquisition of a fluorescence image, the lamp must be switched off. Select the *Adjust* entry in the *Status* list, located in the middle of the dialog box. Use the button showing a small lamp to switch off the lamp.

- The button looks like this, if the lamp is switched off.



- In the *Selected components* list, you can also see that the lamp for this observation method will be switched off.

 Including camera settings

It's usually better to use a black & white camera for acquiring fluorescence images. Should you use a camera that can be toggled between a color mode and a grayscale mode, you can integrate the grayscale mode into the observation mode.

This setting is not necessary if you acquire fluorescence images with the *Multi Channel* acquisition process. Before the *Multi Channel* acquisition process starts, your software checks whether or not your camera is working in the gray scale mode. You will then receive a corresponding message, and can reset the camera before the image acquisition is made.

11. Select your camera in the *Available components* list.
12. Select the *Use* entry in the *Status* list.
13. Some cameras offer gray scale modes in different bit depths. Select the gray scale mode with the highest bit depth from the *Image type* list.
14. Click the *OK* button, to save the new observation method.
 - The *Microscope Control* tool window will then contain a new button with this observation method's name.
 - You can now use this color channel for the *Multi Channel* acquisition process.

Saving the observation method

Using predefined observation methods

As a rule you don't have to completely redefine the observation method required. Use one of the predefined observation methods, and customize it for your microscope.

1. Use the *Acquire > Devices > Device Customization...* command. Activate the *Observation Methods* tab.



- In the *Name* list, you will find all of the observation methods that have been predefined.
Should no observation methods be available, click the *Select Predefined Observation Methods* button. Click the *Select All* button. Click *OK*.
- As soon as filter cubes have been entered for the mirror turret, in your device settings, the appropriate observation methods will be automatically set up. You will always find the observation method beneath the filter cube's name.

2. Select a fluorescence channel (e.g., U-MNU) in the *Name* list.



3. Click the *Rename Observation Method* button.

- The *Enter a New Observation Method Name* dialog box opens.

4. Enter a more general name (e.g., Blue or DAPI), then click *OK*.

- The *Selected components* list contains the following hardware components. There can be more or fewer components, depending on what you have chosen in the way of hardware components in the device list.
- In the mirror turret, the corresponding mirror cube (e.g., U-MNU) will be selected.
- The transmission lamp will be switched off.
- The transmission light path's shutter will have the *Use for acquisition* status. This means that it will be open when the image is acquired.

5. Assign a fluorochrome (such as Blue or DAPI) to the fluorescence channel.

6. Click the *OK* button, to save the new observation method.

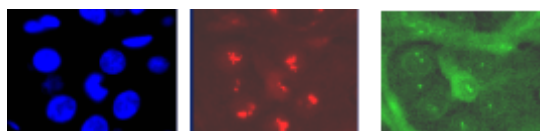
- The *Microscope Control* tool window will then contain a new button with this observation method's name.

00370 29122011

7.3. Acquiring and combining fluorescence image

7.3.1. Acquiring individual fluorescence images

Your software supports several image types. The multi-channel image usually shows a sample that has been stained with several different fluorochromes. However, it is also possible to acquire a multi-channel image that consists only of one single channel.



The illustration shows three fluorescence images of the same sample position. Each image shows another fluorochrome.

Switching to a dark user interface

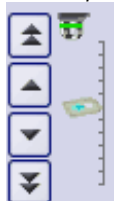
Selecting the fluorescence observation method

Selecting the exposure time for the acquisition




1. If you are disturbed by the light of your monitor, you can switch your software to a dark user interface. To do so, use the *View > Dark Application Skin* command.
2. Use the *View > Toolbars > Observations Methods* command to have the *Observation Methods* toolbar displayed.
3. To load an observation method, click the button with the name of the fluorescence observation method you want.
 - For manual microscopes: Loading a fluorescence observation method defines that a fluorescence image is to be acquired. For your software, all observation methods using the *Fluorochromes* component are automatically identified as fluorescence observation methods.
 - For motorized microscopes: When you load an observation method, this leads to the microscope being brought into a defined condition. To do so, all of the microscope's motorized components will be brought into exactly the position that has been defined for these components in the observation method.
4. Use the *View > Tool Windows > Camera Control* command to make the *Camera Control* tool window appear.
5. Switch to the live mode. To do so, click the *Live* button in the *Camera Control* tool window.
 - For motorized microscopes: The reflected light shutter will be automatically opened. This behavior will be specified when the observation method is defined. For the shutter, the *Use for acquisition* status should have been selected.
6. In the *Camera Control* tool window, select the *Exposure > Manual* option.
7. Some cameras offer the *SFL* mode for fluorescence acquisitions (e.g., the DP72). Switch this mode on.
8. Optimize the exposure time. Should the exposure time become longer than 500 ms, reduce it by increasing gain. To do this, change the value in the *Exposure > Sensitivity* field or use the *Gain* slide control.

Focusing a fluorescence sample



Changing the fluorescence image's display

Saving the fluorescence image

9. Bring the image into focus.
 - If your microscope stage is equipped with a motorized Z-drive, a focus regulator will be at your disposal in the *Microscope Control* tool window.
10. Finish the live mode. To do so, click the *Live* button in the *Camera Control* tool window.
 - For motorized microscopes: The reflected light shutter will be closed.
 - In the image window, you will now see the fluorescence image that has been acquired. The fluorescence image has the image type "Multi-channel image" even if it consists only of one single channel. You can immediately recognize a multi-channel image by this icon  which appears in front of the image name in the document group or in the *Documents* tool window.
 - The fluorescence image will be displayed using the fluorescence color that you have defined together with the observation method.
11. You can use the *Dimension Selector* tool window, to define how the fluorescence image is to be displayed on your monitor, or to change this. There you can, for example, change the fluorescence color.
12. Use the *File > Save As...* command afterwards to save the new multi-channel image. Use the TIF file format.

00395 12032012


7.3.2. Combine Channels

The *Image > Combine Channels...* command creates a new multi-channel image from several separate images. A description of this dialog box can be found in the online help.


Which images can you combine?

 Gray-value images

You can combine a series of gray-value images into a multi-channel image. These can be either 8-bit gray-value images or 16-bit gray-value images. The prerequisite therefore, is that all of the separate images have the same bit depth, image size and image calibration.

 Multi-channel images

Multi-channel images don't necessarily have to be made up of several color channels. There can also be multi-channel images that only contain one fluorescence channel. You can also combine these images into a new multi-channel image that then contains several fluorescence channels. The prerequisite therefore, is that all of the separate images have the same bit depth, image size and image calibration.

 Multi-dimensional images

You can combine several multi-dimensional images into one image. Prerequisite for such an operation is that all of the individual images only differ in one dimension (color channel, focus position, or time point), and are of the same image size.

One example of this is two single color time stacks, that are each made up of 50 separate images. Each time stack was acquired with a different color channel. In this case you can create one multi-channel time stack.

Transmission image

Sometimes another image that shows the same position on the sample in the transmitted light mode, belongs to a series of fluorescence images. You can combine such a transmission image with the multi-channel image.

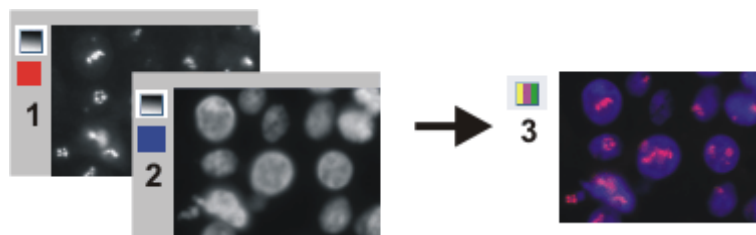
The transmission image doesn't have to be of the same image type as the separate images. However, the image size, image calibration, and the bit depth have to be the same as the fluorescence images.

For example: You can use a true-color image as a transmission image. When the individual fluorescence images have a bit depth of 16 bit, you can only use a 48-bit true-color image as a transmission image.

Combining fluorescence images

Combine Channels

1. Load the gray-value images that you want to combine into a multi channel fluorescence image. The sample was, for example, marked with the fluorochromes, DAPI and Texas Red.
2. Activate the first image in your software's document group.
3. Use the *Image > Combine Channels...* command.
 - The *Combine Channels* dialog box will open.
 - In the *Available Images* table, the active image will be automatically entered as the first color channel.
 - When you have acquired the individual fluorescence images with a suitable observation method, the name of the color channel and the fluorescence color will be read out of the image and automatically used in the *Combine Channels* dialog box.
4. In case you have to change the channel name or want to do so: Click once in the *Name* cell. Enter a description of the color channel, e.g., the name of the fluorochrome used "Texas Red".
You can increase the width of the row so that the description will fit into it.
5. When the fluorescence color can't be read out of the image, the first channel will, by default, be assigned the color 'Red'. To change the active color, click this color field. Select one of the colors from the palette on the *Standard* tab, or activate the *Custom* tab to define a color of your choice.
Click the *OK* button, to close the color palette and return to the *Combine Channels* dialog box.
6. In the next free row, click the *Images* cell. You will be presented with a picklist containing all of the images that you can combine with the active image. Select your next image. As soon as you click in another row, the new image will be included in the sheet.
7. You can now change the characteristics of the new channel. Give the new channel a name and assign a color.
8. It is possible to shift the individual images with respect to each other. To do this, if necessary, use the arrow buttons in the *Pixel shift* group.
9. You can set the weighting of the individual color channels. Increase, e.g., the value in the *Intensity* field, to weight a channel more strongly.
10. Click the *OK* button to create your multi-channel image.
A new image document with the default name "Image_<serial No.>" is created.



With the *Combine Channels* command, you combine fluorescence images (1) and (2) into a multi-channel image (3), this can be done with more than two images also.


Saving a multi-channel image

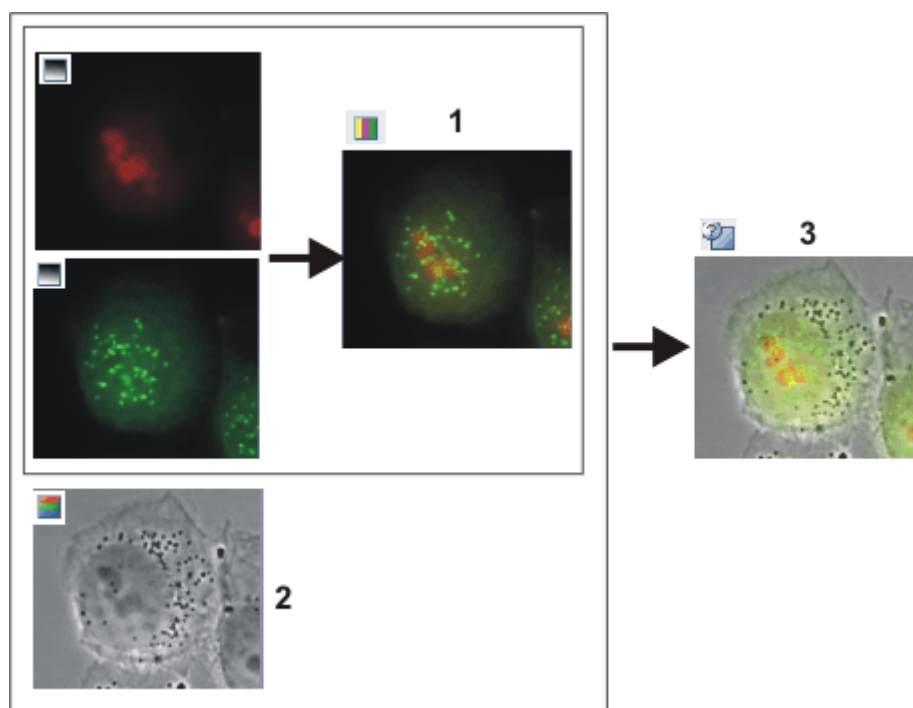
11. Use the *File > Save as...* command to save your new multi-channel image. Always use the TIF or VSI file format when saving an image.

Viewing a multi-channel image

12. Use the *Dimension Selector* tool window, to change the fluorescence color, to choose another color mapping, or to switch individual color channels off and back on.
13. Use the *Adjust Display* tool window, to change the display of a multi-channel image on your monitor. You can e.g., change the weighting of individual color channels in relation to one another.

Combining fluorescence images with a transmission image

1. Load one or more fluorescence images and the transmission image that you want to lay over the color channels.
2. Activate the transmission image in your software's document group.
3. Use the *Image > Combine Channels...* command.
 - The *Combine Channels* dialog box will open.
 - If you want to use a true-color image as a transmission image, that image will be automatically selected in the *Transmission* list. When the transmission image is a gray-value image, it will be automatically entered in the *Available Images* table.
4. If necessary, choose the transmission image in the *Transmission* list.
5. Click once in the *Images* cell. You get a picklist containing all of the loaded images that you can combine with the selected transmission image. Select the fluorescence image you want.
6. If necessary, change the new fluorescence channel's properties. Give the channel a name and assign it a color.
7. Click the *OK* button, to create the resulting image. A new image document with the default name "Image_<serial No.>" is created.
 - In the document group, you can then see a superimposition of all of the images you've combined.
 - The resulting image is a multi-layer image with two image layers. Normally, the two image layers are not of the same image type. For this reason, the image has this icon  in the document group.



With the *Combine Channels* command, you combine several fluorescence images into a multi-channel image (1). If you've acquired the transmission image at the same place on the sample (2), you can combine it with the multi-channel image to make a multi-layer image (3).

Viewing a multi-layer image



8. Click this button in the navigation bar to hide the transmission image.

- Now, you will only see the multi-channel fluorescence image.



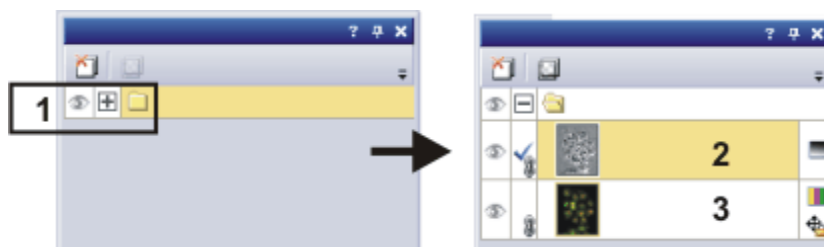
9. Click this button and show the transmission image again.


- Now, you see a superimposition of the transmission image and the multi-channel fluorescence image.

Moving the transmission image with respect to the multi-channel image

10. Use the *View > Tool Windows > Layers* command to make the *Layers* tool window appear.

11. In the *Layers* tool window, click the [+] sign (1) and open the image's layers.



- You can now see the image's individual layers: transmission image (2) and multi-channel image (3). The height map can't be seen, because it's absolutely transparent at the moment. The icon  at the left side of the multi-channel image means that it is not possible to move the multi-channel image.

12. Select the transmission image in the *Layers* tool window.

13. Activate the *Toolbox* toolbar. To do this, use, e.g., the *View > Toolbars > Toolbox* command.



14. Click the *Move Tool* button on the *Toolbox* toolbar.

- On the image window, the mouse pointer will change its shape.

15. Move the whole image with the left mouse button depressed.



16. Click e.g., the *Selection Tool* button on the *Toolbox* toolbar to leave the move mode.

Changing the weighting between a transmission image and a multi-channel image

When you display the transmission image and the multi-channel image simultaneously in the image window, the transmission image will cover the multi-channel image, and for this reason, you can't see the multi-channel image. You can display both images transparently, and in that way be able to see parts of both images.

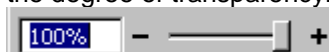
17. To start with, select the image layer in the *Layers* tool window. To do so, simply click the layer's name.

- The layer you have selected will then be shown with a colored background in the tool window.



18. Then click the *Set Layer Opacity* button. You can find this button in the tool window's toolbar.

- In the tool window a slide control will then appear, with which you can set the degree of transparency.



19. Use the slide control to set the degree of transparency you want. At a value of 100% the image layer is opaque. At a value of 0% the image layer will be completely faded out.

20. When you're satisfied with the transparency setting, click once on any place on the user interface.

Saving a multi-layer image

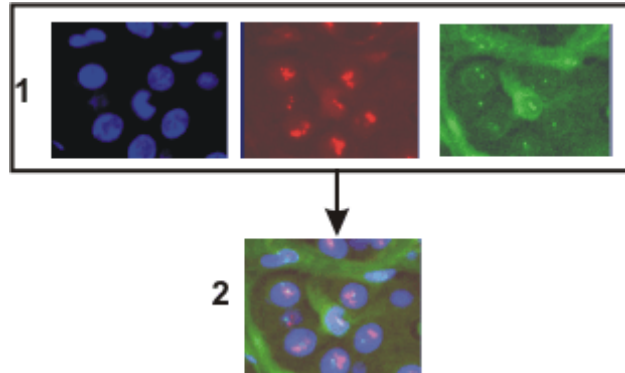
21. Use the *File > Save as...* command, to save your new multi-layer image. Always use the TIF or VSI file format when saving an image.

00067 25072011

7.4. Acquiring multi-channel fluorescence images

Use the *Multi Channel* automatic acquisition process to acquire a multi-channel fluorescence image.

Example: Define a process for the acquisition of a multi-channel fluorescence image (e.g., Blue, Green, and Red). When you set up the fluorescence sample, illuminate it as little as possible to minimize the bleaching effect.



At the top of the illustration you can see the individual fluorescence images (1). Below, you can see the superimposition (2) of the separate fluorescence images.

Defining the "Multi Channel" acquisition process

Selecting the acquisition process

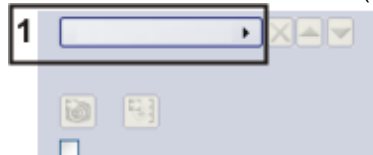


1. Use the *View > Tool Windows > Process Manager* command to make the *Process Manager* tool window appear.
2. Select the *Automatic Processes* option.
3. Click the *Multi Channel* button.
 - The button will appear clicked. You can recognize this status by the button's colored background.
 - The [C] group will be automatically displayed in the tool window.
4. Should another acquisition process be active, e.g., *Z-stack*, click the button to switch off the acquisition process.
 - The group with the various acquisition processes should, for example, now look like this:



Adding color channels

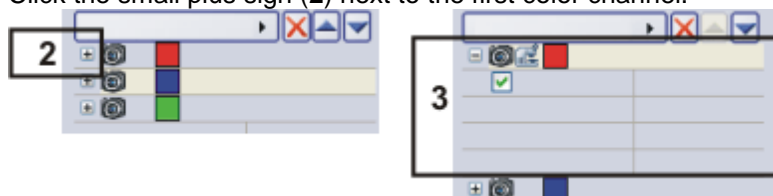
5. Click the *Add Channel* button (1).



- A context menu will open. The context menu will list all of the observation methods that are currently defined.
6. Select the color channel that is to be acquired first, e.g., Red.

Selecting the exposure times for the individual color channels

7. Select the other channels (e.g., Green and Blue) in the same manner. Note that the fluorescence images are later on acquired in the same order as they are listed there.
8. Use the *View > Tool Windows > Camera Control* command to make the *Camera Control* tool window appear.
9. Click the small plus sign (2) next to the first color channel.



- The channel has now been activated (3). The active color channel will be shown highlighted in color in the tool window.
- The color channel entries in the *Process Manager* tool window are organized like a tree view. Expand an entry to open a list with additional information about the selected color channel.
- When you activate the color channel, you also automatically select the corresponding observation method. You can recognize which observation method is active, by the microscope icon. At the same time, this means that the microscope is now set up correctly for the acquisition of the fluorescence image for the first color channel.



10. Switch to the live mode. To do so, click the *Live* button in the *Camera Control* tool window.
 - The reflected light shutter will be automatically opened. This behavior will be specified when the observation method is defined. For the shutter, the *Use for acquisition* status should have been selected.
11. In the *Camera Control* tool window, select the *Exposure > Manual* option.
12. Some cameras offer the *SFL* mode for fluorescence acquisitions (e.g., the DP72). Switch this mode on.
13. Optimize the exposure time. Should the exposure time become longer than 500 ms, reduce it by increasing gain. To do this, change the value in the *Exposure > Sensitivity* field or use the *Gain* slide control.



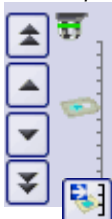
14. In the *Process Manager* tool window, click the *Read settings* button.



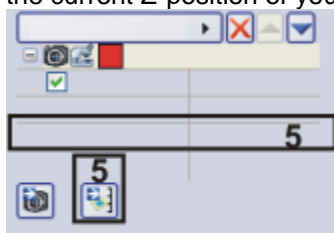
- This exposure time will be automatically adopted for the active channel, and will be shown in the *Process Manager* tool window (4).
15. Then, activate the other channels and set the exposure time for each of them.
 16. Finish the live mode. To do so, click the *Live* button in the *Camera Control* tool window.
 - The reflected light shutter will be closed.



Focusing a
fluorescence sample




17. Activate the first channel.
18. Switch to the live mode.
19. Bring the image into focus.
 - If your microscope stage is equipped with a motorized Z-drive, a focus regulator will be at your disposal in the *Microscope Control* tool window.
20. Click the *Read Z-offset* button in the *Process Manager* tool window to adopt the current Z-position of your microscope stage (5).



21. Activate the other channels, focus the sample and transfer the current Z-position of the microscope stage to the acquisition process.
 - The first color channel is always used as a reference for the Z-offset. Below the other color channels you can find the *Z-offset* value. It shows how the focus positions of the individual color channels differ from each other.
22. Select the *Use Z-offset* check box (6).



Finishing the process
definition

23. Finish the live mode.
 - The reflected light shutter will be closed.
24. Click the *Save Process Definition*  button in the toolbar at the top of the *Process Manager* tool window to save the acquisition parameters for the process that has just been defined.
 - A channel contains an observation method, an exposure time, a gain value, and where necessary, a focus position. All of these settings will be saved together with the process definition.
 - You can now use the acquisition parameter for this acquisition process again, at any time.

Acquiring and viewing a multi-channel fluorescence image

Defining the acquisition process

1. Define a process for the acquisition of a multi-channel fluorescence image, or load acquisition parameters that have already been saved. To do this, click the *Load Process Definition* button, located in the *Process Manager* tool window's toolbar.

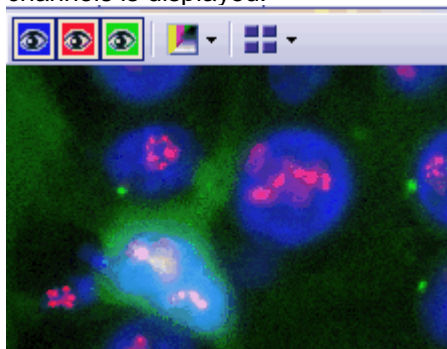
Switching to a dark user interface

2. If you are disturbed by the light of your monitor, you can switch your software to a dark user interface. To do so, use the *View > Dark Application Skin* command.

Starting the acquisition process



3. In the *Process Manager* tool window, click the *Start* button.
 - If you use a manual microscope, you'll receive several different messages about switching the mirror cube and opening and closing the shutter.
4. For microscopes that aren't motorized: Follow the instructions and make the necessary settings on your microscope.
 - The acquisition of the multi-channel fluorescence image starts immediately. The order of the color channels in the *Process Manager* tool window corresponds to the order in which the color channels were acquired.
 - The acquisition has been completed when you can again see the *Start* button in the *Process Manager* tool window.
 - The multi-channel fluorescence image will be automatically saved. You can set the storage directory in the *Acquisition Settings > Saving > Process Manager* dialog box. The preset file format is VSI.
 - In the image window, the superimposed fluorescence image of all channels is displayed.

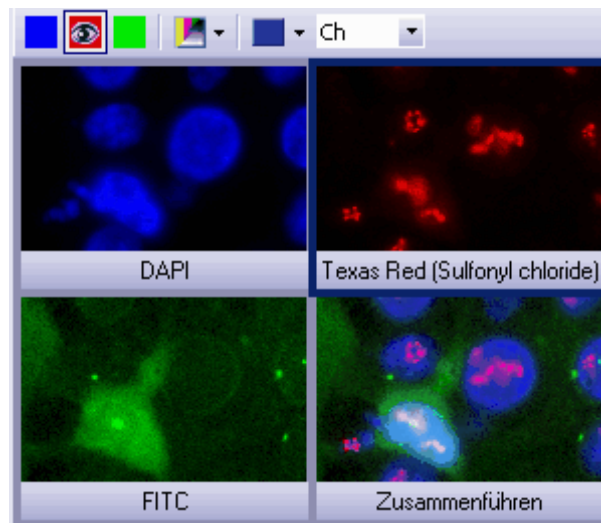



Viewing a multi-channel image



- The navigation bar will be displayed at the top of the image window. It contains a button for each channel to enable you to display or hide that channel. The eye icon indicates that the channel is currently visible.
5. Click the color channel button in the navigation bar to have a color channel displayed or hidden. Take a look at all of the color channels one by one.
 6. When you've finished, superimpose all of the channels again.
 7. Click the *Tile View* button located in the navigation bar to change the image window view.
 - You will then see all of the color channels that have been acquired.





8. Compare the color channels.
9. Click the *Single Frame View* button on the navigation bar.
 - You will then once more see the superimposition of all of the color channels in the image window.
10. Use the *View > Tool Windows > Properties* command to make the *Properties* tool window appear.
 - In the *Properties* tool window, you can find that every color channel has its own *Channel* information group.
11. Should an information group has been reduced: Click the plus sign  to have all of the information displayed again.
 - The color channel's name, the corresponding wavelength, the observation method, and the exposure time will all be shown for each color channel.

Viewing information on
the individual color
channels

Acquiring a multi-channel fluorescence image together with a transmitted light image

Task Simultaneously with the multi-channel fluorescence image, acquire a transmitted light image, e.g., a phase contrast image.

Defining the acquisition process

Adding the acquisition of a transmitted light image to the acquisition process

Defining the transmitted light image acquisition



Starting the acquisition process



1. Define an acquisition process for a multi-channel fluorescence image. To do this, follow the step-by-step instructions described above.
2. Click the [Add Channel](#) button. Choose an observation method for the acquisition of a transmitted light image, e.g., phase contrast, differential interference contrast (DIC), or brightfield.
3. Click on the transmitted light channel in the [Process Manager](#) tool window.
 - The channel has now been activated. Your microscope will be set in the transmitted light mode.
4. Click the small plus sign next to the transmitted light channel.
 - You'll now see a table with additional information about the transmitted light channel.
5. Make sure that the [Transmission overlay](#) check box has been selected. Only then is the transmitted light image assigned its own layer that lies over the fluorescence channels.
6. Switch to the live mode.
7. Select manual exposure time in the [Camera Control](#) tool window. In order to set the sensitivity to the lowest ISO value, set the gain to the value of 0. Optimize the exposure time.
8. In the [Process Manager](#) tool window, click the [Read settings](#) button.
 - The exposure time will be adopted for the channel.
9. Finish the live mode.
10. In the [Process Manager](#) tool window, click the [Start](#) button.
 - Then, together with your fluorescence images, a transmitted light image will also be acquired and saved together with the multi-channel fluorescence image. The result of this acquisition process is a multi-layer image that you can view with the [Layers](#) tool window.
11. Take a look at the multi-channel fluorescence image with the superimposed transmitted light image in the image window.

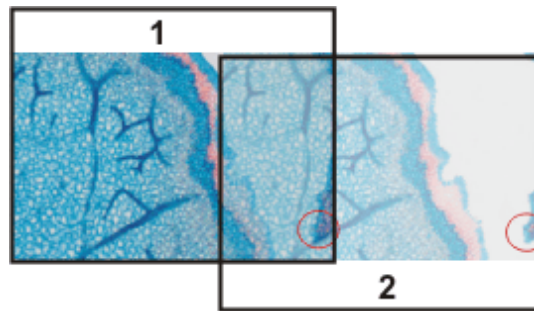
00369

8. Creating stitched images

Acquiring a stitched image without a motorized XY-stage (Manual MIA)

Example: You want to acquire an image of a large sample area. Use the *Manual MIA* acquisition process to acquire several individual images of adjoining positions on the sample, and to have them combined into a stitched image.

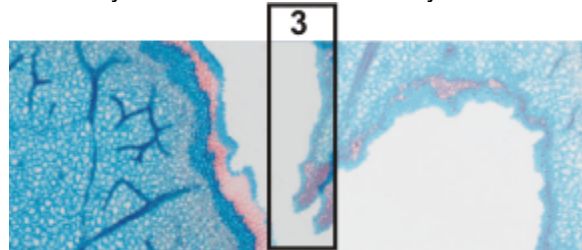
- Prerequisite* The camera is aligned parallel to the XY-stage. The angle between camera and stage should be smaller than 1°.
- Selecting an objective*
1. On the *Microscope Control* toolbar, click the button with the objective that you want to use for the acquisition of the stitched image.
- Setting the image quality*
1. Switch to the live mode, and select the optimal settings for your acquisition, in the *Camera Control* tool window. Pay special attention to setting the correct exposure time. This exposure time will be used for all of the stitched image's individual images.
 2. Find the position on the sample at which you want to start acquiring the stitched image.
 3. Finish the live mode.
- Selecting the acquisition process*
1. Activate the *Process Manager* tool window.
 2. Select the *Manual Processes* option.
 3. Click the *Manual MIA* button.
 - The button will appear clicked. You can recognize this status by the button's colored background.
 - The *Manual MIA* group will be automatically displayed in the tool window.
 - Should the *Instant EFI* acquisition process have been active, it will be automatically switched off. You can, however, use images with extended depth of focus for the stitched image. To do this, before you acquire each of the individual images, click the *Instant EFI* button located in the *Manual MIA* group.
- Selecting the acquisition parameters*
1. Make quite certain that the *Auto Align* button appears clicked. It should then look like this:
 - Then your software will search for the same image structures in neighboring individual images. The stitched image will be put together in such a way that image areas that are the same will be superimposed.
- Acquiring a stitched image*
1. Click the *Start* button.
 - Your software switches into the live mode.
 2. Bring the sample into focus.
 3. Click on one of the arrow buttons to set the side of the current image at which the next image is to be arranged. For example, click this button if the next image is to be laid to the right of the current image.
 - Your system now acquires an image at the current position on the sample. In the image window you now see on the left (1) the acquired image, and on the right (2) the live-image is displayed.




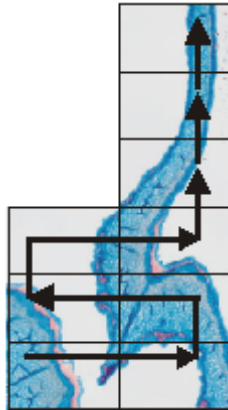
Since you haven't moved the sample, the live-image still shows the current sample position, too, which means that you now see the current image twice.

The two images overlap. Since the live-image is shown transparent, you see both images in the overlap area simultaneously.

4. Make a note of a significant structure on the live-image's right border. You will find the same sample structure in the overlap area. On the illustration, a significant structure has been indicated by a circle.
5. Now move the stage very slowly to make the structure on the live-image move to the left. Keep moving the stage until the image structures in the overlap area lie as exactly over each other as possible. The image structures need not lie precisely over each other, since your software will match the individual images with each other.
 - In the overlap area (3), the same image segments are shown now. This enables your software to seamlessly combine the two images.

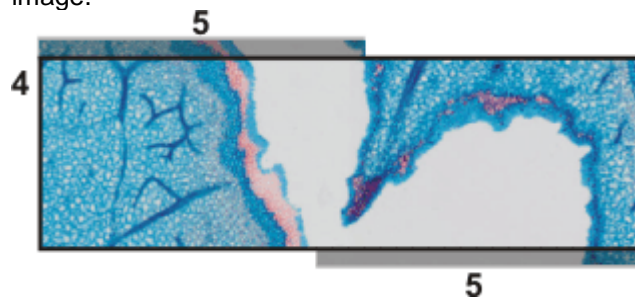


- You can reverse the direction in which your stage moves, in the [Device Settings > Stage](#) dialog box. Depending on how you can best orient yourself, the live-image will then move to the left or to the right, when you move your stage to the right.
6.  Check whether both images have been correctly combined. Otherwise, you can undo the last step by using the [Undo last frame](#) button. You can then move the stage again, and match the structures better.
 - During the acquisition, you can change the current stitched image's zoom factor, e.g., to see certain parts in the overlap area better. You will find an overview on the possibilities of changing an image's zoom factor in the online help.
 7. Define your way through the sample, with the arrow buttons, and follow that with the stage.
In this manner, you can display a sample in any form you like in the stitched image. The illustration shows a stitched image that is made up of 9 individual images, and the stage path.



8. Click the *Stop* button when you want to end the acquisition of the stitched image.

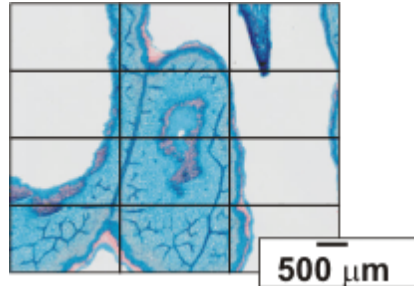
- You see the completed stitched image (4) in the image window. Since the individual images can lie a little askew of each other, the stitched image isn't as a rule, rectangular, but contains empty areas on its borders (5). These areas will, as a rule, be cut off in the stitched image.



- The stitched image will, by default, be automatically saved. The storage directory is shown in the *Acquisition Settings > Saving > Process Manager* dialog box. The preset file format is VSI.
- By default, in the overlap area, the intensity values of two adjoining individual images will be matched with each other to make the image's overall impression homogeneous.
- Stitched images are calibrated. This means that you can measure distances and objects on a stitched image.

Properties of the stitched image

Acquiring a stitched image with a motorized XY-stage (XY-Positions / MIA)



Example: You want to acquire an image of a large sample area. Use the automatic *MIA* acquisition process, to scan a rectangular area of the sample and to have adjoining images combined into one stitched image.

Preconditions

- The stage has been set up and initialized, i.e. its stage limits have been defined.
- The camera is aligned parallel to the XY-stage. The angle between camera and stage should be smaller than 1°.
- The shading correction has been set up.

Selecting the acquisition process



1. Activate the *Process Manager* tool window.
2. Select the *Automatic Processes* option.
3. Click the *XY-Positions / MIA* button.
 - The *XY* group will be automatically displayed in the tool window.

Using the software autofocus



1. If your microscope is equipped with a motorized Z-drive, you can switch on a software autofocus.
In the *Process Manager* tool window, click the *Autofocus* button.
 - The button will appear clicked. You can recognize this status by the button's colored background.
 - The *Autofocus* group will be automatically displayed in the tool window.
2. In the *Autofocus* group, select the *Multiposition / MIA* check box.
3. If the sample surface is not plane or if it is inclined to the objective, choose the *Every MIA frame* option. Now, the software autofocus will be performed before every image acquisition.

Putting the stage navigator on display



1. In the *Process Manager* tool window, click this button.
 - The *Stage Navigator* tool window will be shown. When you have acquired an overview image of your sample, you will see this area of the image in the stage navigator's image segment.
2. Set the magnification for the image segment in the *Stage Navigator* tool window. To do this, use the zoom buttons at the bottom left of the tool window (2).
The current stage position will be shown by a yellow rectangle in the image segment (1). You should choose a magnification that enables you to see this rectangle clearly.

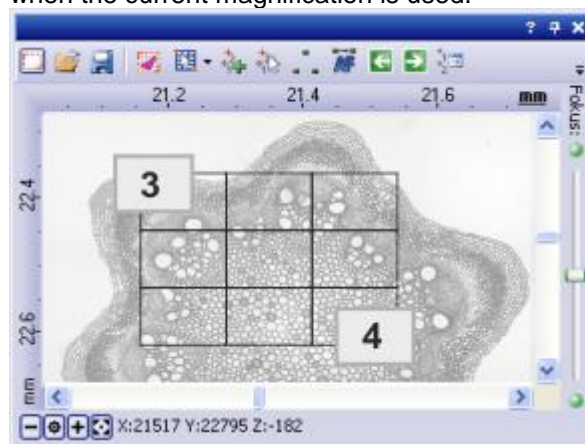


- You can find more information on the [Stage Navigator](#) tool window in the online help.

Defining the MIA scan area



1. In the [Process Manager](#) tool window, click this button.
 - The system will automatically switch into the live mode.
 - The [Define MIA Scanning Area](#) dialog box opens.
2. Move the XY-stage to the top left-hand corner of the MIA scan area you want (3).
3. Focus, then select the optimal settings for your acquisition, in the [Camera Control](#) tool window. Pay special attention to setting the correct exposure time. This exposure time will be used for all of the stitched image's individual images.
4. Confirm the starting position in the [Define MIA Scanning Area](#) dialog box with **OK**.
5. Move the XY-stage to the bottom right-hand corner of the MIA scan area (4). Confirm this position in the [Define MIA Scanning Area](#) dialog box with **OK**.
 - In the [Stage Navigator](#) tool window, the MIA scan areas that have been defined are displayed. Here, you can immediately see how many individual images are required for the acquisition of the stitched image, when the current magnification is used.



Acquiring a stitched image

1. Click the *Start* button.
 - The acquisition begins immediately. The individual images are acquired, then immediately assembled. You can watch how the stitched image grows, in the image window.
 - You see the completed stitched image, in the image window. The individual images won't be saved separately.
 - The stitched image will, by default, be automatically saved. The storage directory is shown in the *Acquisition Settings > Saving > Process Manager* dialog box. The preset file format is VSI.

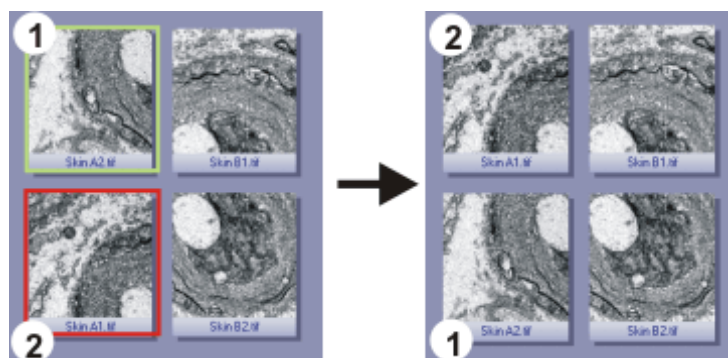
Combining individual images into a stitched image

Loading and selecting images

1. Load the images you want to combine, or acquire a suitable set of images.
2. Open the *Gallery* tool window. To do this, use, e.g., the *View > Tool Windows > Gallery* command.
3. Select all of the images you want to combine, in the *Gallery* tool window.

Assembling images

1. Use the *Process > Multiple Image Alignment...* command. This command is only active when more than one image of the same image type has been selected.
 - The *Multiple Image Alignment* dialog box will open. A description of this dialog box can be found in the online help.
 - The dialog box's stitching area will display a preview of the individual images.
2. If necessary, while keeping your left mouse button depressed, drag on the bottom left-hand corner of the dialog window to enlarge it. Alternatively, double click the header of the dialog box to enlarge the dialog box to full-screen size.
3. Check whether the images' positions are correct. You can change the arrangement of the individual images, e.g., by exchanging two images in the stitching area by Drag&Drop.



- The illustration shows the stitching area with four individual images. On the left, the images 1 and 2 are not in the correct position. Image 1 (green frame) will therefore be dragged onto image 2 (red frame). On the right, you see the stitching area after the two images have been interchanged.
4. When the individual images overlap, select the *Correlation* option in the *Output > Alignment* list. Then your software will search for the same image structures in neighboring individual images. The stitched image will be put together in such a way that image areas that are the same will be superimposed.

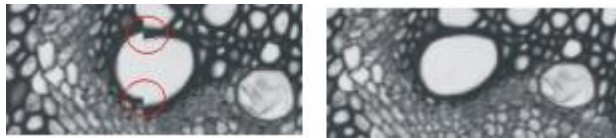
Checking a stitched image

5. Click the *OK* button to carry out the automatic image alignment.
 - The *Multiple Image Alignment - Manual Align* dialog box opens.
 - The stitched image will be displayed.
1. Check the stitched image on display. Use the zoom buttons in the dialog box to zoom in the stitched image in the dialog box.



2. Should individual images have been incorrectly assembled, you can manually shift one or more of them, in respect to one another. To do this, click in the image you want to shift, then drag it with your left mouse button depressed, in the required direction.

- The currently selected image will be displayed semi-transparently to make it easier for you to find the point of contact with the neighboring image.




- Two images were not correctly aligned with each other. There is a misalignment. When the manual alignment has been made, the two images fit together seamlessly.
3. Select the *Cut Edges* check box to clip the image in such a way that there are no longer any empty areas visible on its borders.
 - In the preview, the image edges that are to be clipped will be displayed semi-transparently.
 4. Select the *Equalize* check box if the images aren't homogeneously illuminated. Then the intensity values of the individual images will be matched with one another, which will make the background appear more homogeneous.
 5. Click *OK*.
 - A new image with the name "Image_<consecutive No.>" will be created.

00383

9. Processing images

The *Process* menu offers numerous image processing functions, with which you can change an acquired image (e.g., increase the image contrast or the image sharpness).

1. Load the image you want to process, or activate the image in the document group.
 - Please note that the *Process* menu will only be visible when an image window is active in the document group.
2. Use one of the commands in the *Process* menu, e.g., *Process > Enhancement > Adjust Intensity...* .
 - The image processing dialog box will open. The image processing operation that is active will be shown in the dialog boxes header.
- 
 3. Click the small arrow next to the *Preview* button to open a list of all of the preview functions. Select the *Original and Preview* entry.
 - This preview function displays the same image segment twice in the dialog box. The first one shown is the source image. The second is the image that results when the current parameters are used.
 - Most of the image processing operations need one or two of the parameters that are shown in the *Settings* group.
4. Change the image processing operation's parameters. After every change that is made in a parameter, the operation will be immediately applied to the source image, and the resulting image will be shown in the preview window. Click the *Default* button, to readopt the preset parameters in the *Settings* group, when the current parameter doesn't make sense to you.
5. When you have found the optimal parameters, click the *OK* button to have the active image processing operation applied to the image with the active parameters.
 - The image processing dialog box will closed.
 - Please note that the image processing operation changes the source image. No new image document will be created. You can, however use the *Edit > Undo* command to restore the source image.












00175 22022011

10. Life Science Applications

The *Life Science Application* toolbar offers you various evaluation methods for your images. If this toolbar is not displayed, use the *View > Toolbars > Live Science Applications* command.



The following table lists the buttons which are available by default on the toolbar.

	New ROIs	Use one of several options to define an image segment in the active image as a region of interest (ROI).
	Intensity profile	An intensity profile shows how the intensity within one, or within several image segments (ROIs), changes over a period of time or over the different Z-positions.
	Fluorescence Unmixing	Use fluorescence unmixing, to remove spectral mixing from a multi-channel fluorescence image.
	Brightfield Unmixing	Use brightfield unmixing, to break down a brightfield image containing three different colors into its individual color components.
	Colocalization	On a multi-channel fluorescence image, measure the colocalization to identify image segments where the individual fluorescences overlap.
	Ratio Analysis	Measure how the calcium ions concentration is changing in a time stack.
	Verify Channel Parameters	For the active image, check whether all of the parameters have been correctly defined which are necessary for successful deconvolution.
	2D deconvolution	Use a deconvolution filter to remove disturbing diffused light from an image.
	Nearest Neighbor	
	Wiener	
	Constrained Iterative Filter	

6619 16072013

10.1. Intensity profile

With the *Measure > Intensity Profile...* command you can measure the intensity profile over the time (time stack) or over the different Z-positions (Z-stack). An image series can be a time stack or a Z-stack.

As a result you will obtain an intensity profile that shows how the intensity within one, or within several image segments, changes over a period of time or over the different Z-positions.

Example of use You can use intensity profiles to measure how concentrations change with time. For example, when you make experiments with triggering the calcium flow with ATR, and use suitable fluorescence stains.

Before using the command To calculate an intensity profile, all of the pixels within a specific image segment will be evaluated. Your software will, e.g., determine the mean intensity of all of the pixels.

Before you can measure an intensity profile, you have to define this image segment. To do this, define one or more ROIs (Regions Of Interest) on the image. To define these ROIs, you can, for example, use the buttons on the *Life Science Applications* toolbar. You can find more information on working with ROIs in the online help.

Supported image types With the *Measure > Intensity Profile...* command you can measure the following image types:



Time stacks, whose frames are gray-value images.



Z-Stacks, whose frames are gray-value images.



Multi-channel Z-stacks, multi-channel time stacks

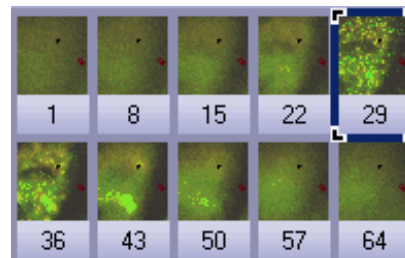
Prerequisite: The command is only available for gray-value images. Use the *Image > Mode > Grayscale* command, to convert the an image into a gray-value image.

00324

Measuring intensity profiles

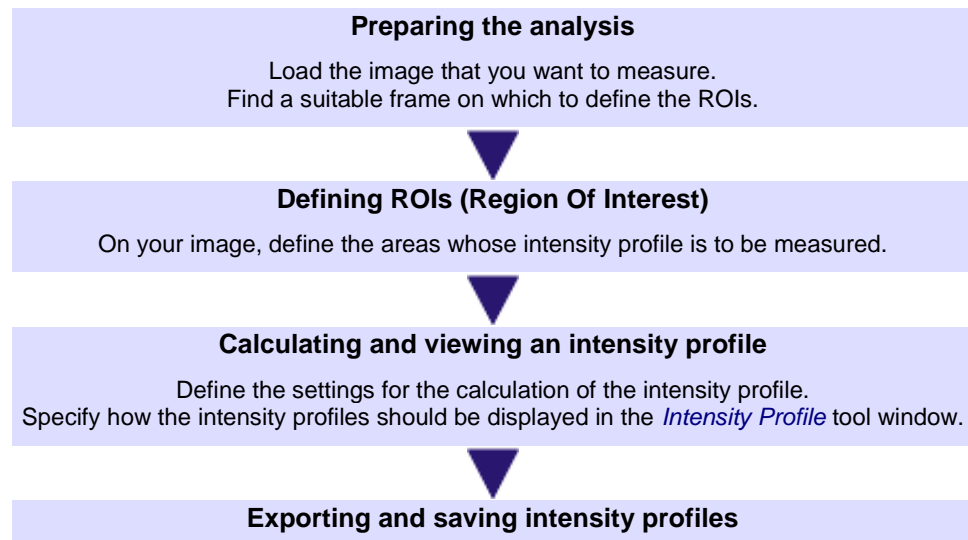
With the *Measure > Intensity Profile...* command, you can measure the intensity profile over the time (time stack) or over the different Z-positions (Z-stack).

Task You have acquired a focus series for several fluorescences. You want to know how the intensity develops at a variety of positions on the sample at a variety of Z-positions.



The illustration shows an overview over the frames in a multi-channel Z-stack. The multi-channel image contains a red and a blue color channel. For the acquisition of the Z-stack a through-focus series was taken of the sample. The sample can only be seen clearly, and sharply focused, in the middle of the Z-stack.

The following process flow chart displays the basic steps of the process.



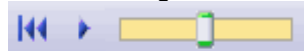
Preparing the analysis

- Several example images were supplied together with your software. You can follow these step-by-step instructions using the PeroxysomOrganelles.tif example image. This example image is a multi-channel Z-stack image.

Displaying a suitable image for the definition of the image segment

- When you load a multi-channel Z-stack, it will be automatically displayed in the Single Frame View in the image window.

- Use the navigation bar at the top of the image window.



- Move the slide control slowly, and by doing so display frames acquired at differing Z-positions in the image window. Search out a Z-position at which the sample can be clearly recognized.

- Use the *View > Toolbars > Life Science Application* command, to have the *Life Science Application* toolbar displayed. You can find the functions for defining ROIs and for measuring the intensity profile on this toolbar.

Defining ROIs (Region Of Interest)

- Rotate the mouse wheel to change the zoom factor. Enlarge the image until you can see at least one enlarged segment of the sample in the image window, that is fluorescing in red.

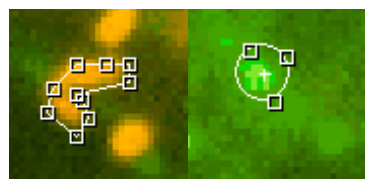


- Click the *New ROI - Polygon* button on the *Life Science Applications* toolbar.

- By clicking with your left mouse button, define an area on the image that only includes red fluorescing sample positions.

- Rightclick to finish the definition of the ROI.

- Then define another ROI on an image segment that only includes green fluorescing sample positions.



- Click the *New ROI - Rectangle* button.

- Define a square in a dark image segment that shows no fluorescing objects. This ROI will be used as a reference for the background correction.

- On the *Life Science Applications* toolbar, click the *Intensity Profile* button.

Calculating an intensity profile



- The *Intensity Profile - <Name of the active image>* dialog box opens.
 - Your software recognizes the image type, and automatically selects the corresponding option. In this example the *Z-profile* option is preset.
13. Select the *Results > Average* check box. Clear all the other check boxes.
 - In the *ROI data* group, all of the ROIs that have been defined on the active image will be listed. In this example, you'll find three ROIs there (two on sample positions showing different fluorescence colors and one on the background).
 14. Each ROI defines a specific image segment. Now, select the image segments for which intensity profiles are to be calculated. In this example, select both of the ROIs at fluorescing sample positions.
 15. In the *Background Subtraction* group, select the *ROI* option.
 - In the list next to the *ROI* option, all of the ROIs that have been defined on the active image will be listed.
 16. In the list, select the ROI that was defined on the image background.
 17. Click the *Execute* button.
 - The intensity profiles will be calculated and displayed in the *Intensity Profile* tool window.

Viewing intensity profiles



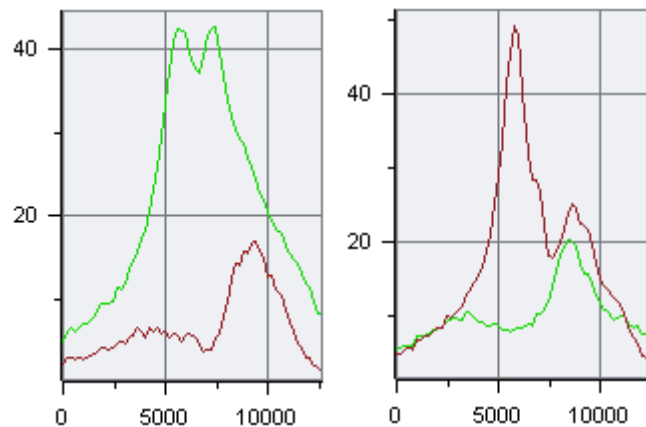
18. If necessary, use the *View > Tool Windows > Intensity Profile* command, to show the tool window. The tool window offers you several ways of displaying the intensity profile that has been measured.
19. In the *Intensity Profile* tool window's toolbar, click the *Arrange intensity profile charts* button.
 - The *Arrange intensity profile charts* dialog box opens.
20. Make the following settings in the dialog box. Select the *Select all* check box, in the *Show charts* group. In the *Layout* group, specify a grid size of 2x1.



Close the dialog box with *OK*.



21. In the *Intensity Profile* tool window's toolbar, click the *Arrange intensity profile data* button.
 - The *Arrange intensity profile data* dialog box opens.
22. Make the following settings in the dialog box. Select the *Separate chart per channel* check box. Clear the other check boxes. Close the dialog box with *OK*.
 - You can see two charts, each with two curves. Along the X-axis the Z-position, that's to say, the height, has been plotted. The intensity range has been plotted along the Y-axis.



For each of the image's color channels, an individual chart will be created. The name of the corresponding color channel will be displayed in the chart's header. On the left, you see the results for the green color channel, on the right, those for the red one.

In each chart, you will see a curve for each ROI that has been defined. You can display a legend with the name of the ROIs in the chart. The green curve was measured on the ROI on the green fluorescing position on the sample, the red on the red fluorescing position.

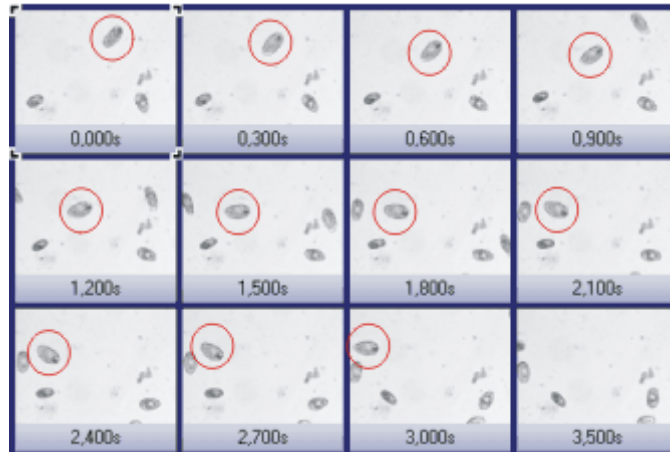
Exporting and saving
intensity profiles



23. In the *Intensity Profile* tool window's toolbar, click the *Export to Workbook* button.
 - A new workbook will be created in the document window. This workbook contains results sheets with all of the results. When you've measured a multi-channel image, you'll find an individual work sheet for each of the color channels.
24. Use the *File > Save as...* command, to save a workbook.
 - A workbook will be saved in the file format OWB. This format is an exclusive file format and can only be opened with your software. Workbooks are, obviously, therefore not suitable for using to exchange data with other application programs. If you would like to use the results in a different application, use the *File > Export to > Excel...* command.

Measuring the intensity profile of moving objects

Task You've acquired a time stack of moving paramecia. Define a dynamic ROI that contains a paramecium and move the ROI so that it follows the paramecium through all of the frames in the time stack. Measure the intensity profile.



The illustration shows an overview over the frames in a Z-stack. The time points associated with the frames are shown under the images. The red circle shows the movement of a single paramecium.

Specifying the user interface and default settings

- Several example images were supplied together with your software. You can follow these step-by-step instructions using the `ParameciumTimeSeries.tif` example image.
- Use the **View > Toolbars > Life Science Application** command, to have the *Life Science Application* toolbar displayed. You can find the functions for defining ROIs and for measuring the intensity profile on this toolbar.
- If necessary, use the **View > Tool Windows > Measurement and ROI** command to have the *Measurement and ROI* tool window displayed. The ROIs that are defined in the current image are listed in this tool window.
- Use the **Tools > Options...** command. Select the *Measurement and ROI > Dynamic ROI* entry in the tree view. Select the *Interpolate linearly, continue with current* option. You have now defined how a dynamic ROI behaves when you define its position on the frames. Close the dialog box with **OK**.

Viewing the movement of the paramecia

- Use the navigation bar at the top of the image window.



- Move the slide control slowly to the right to view the movement of the paramecium that is at the top left border of the image in the first frame.
 - The paramecium first moves down and to the left. Then it changes direction and moves up before finally disappearing at the left border of the image.
- Display the first frame in the image window. To do this, use the navigation bar at the top of the image window.

Defining ROIs (Region Of Interest)



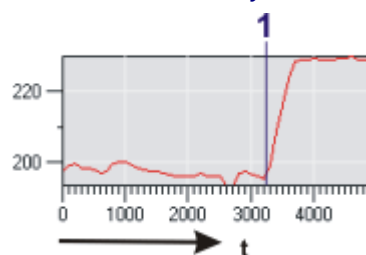
- Click the **New ROI - Rectangle** button on the *Life Science Applications* toolbar.
 - The ROI is displayed in the sheet of the *Measurement and ROI* tool window. In the *Type* column, the keyword *(ROI)* is added to the type name.



Following the movement of the object using the dynamic ROI

9. With two mouse clicks, define a small rectangle around the paramecium at the top left border of the image.
10. Move the mouse pointer over the ROI you just defined. Click the right mouse button to open a context menu. Select the *Convert to dynamic ROI over t* command from the context menu to turn the static ROI into a dynamic ROI.
 - In the *Measurement and ROI* tool window, the keyword (*ROI*) in the *Type* column changes into the new keyword (*dROI [t]*).
11. In the image window, display the frame in which the paramecium changes its direction.
 - The position of the ROI that has been defined is the same on all frames.
12. Move the ROI on this frame so that it contains the paramecium again.
 - The system will now automatically reposition the ROI on each frame between the first and the current frame. The positions are calculated as a linear interpolation of the ROI positions in the first and the current frames. Check whether the paramecium is completely within the ROIs on these frames.
13. In the image window, display the last frame in which the paramecium is still completely visible in the image.
14. Move the ROI on this frame again so that it contains the paramecium again. Make sure that the ROI doesn't include any other paramecium.
15. Check whether the ROIs position is correct for all of the frames up till now.
 - In the following frames, the paramecium disappears at the left border of the image and so can't be measured any more. The dynamic ROI is still defined on all following frames in the image series. You can't delete a dynamic ROI only for particular frames.
16. On the *Life Science Applications* toolbar, click the *Intensity Profile* button.
 - The *Intensity Profile - <Name of the active image>* dialog box opens.
 - Your software will recognize the image type, and will select the appropriate option in the *Method* group. In this example the *Over time* option is preset.
17. Make the following settings in the *Intensity Profile* dialog box.
 - Select the *Results > Average* check box.
 - Clear the other check boxes.
 - Select the dynamic ROI in the *ROI data* group.
 - In the *Background subtraction* group, select the *none* option.
18. Click the *Execute* button.
 - The intensity profile for the paramecium will be calculated and displayed in the *Intensity Profile* tool window.

Calculating an intensity profile

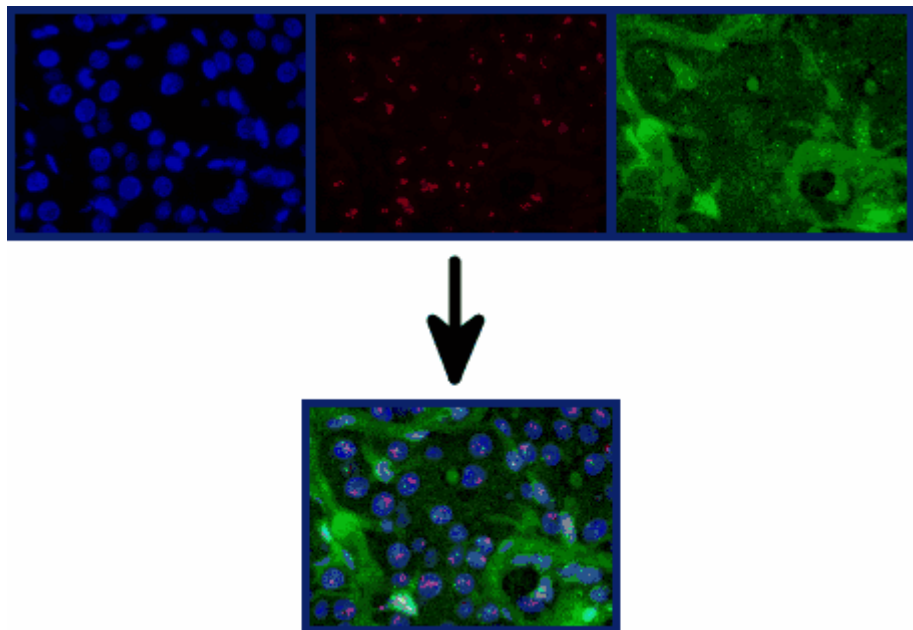


The intensity profile displays how the average intensity in the ROI changes over time. The ROI contains the paramecium until about 3000 ms. The intensity is relatively constant. At time point (1), the paramecium begins to leave the image. The intensity then increases to the level of the light image background. The time point is at about 3300 ms.

10.2. Fluorescence Unmixing

Multi-channel fluorescence microscopy

In the multi-channel fluorescence microscopy, different cell structures will be visually separated by acquiring them separately, then displaying them in different colors. To achieve this, one stains the sample with several suitably chosen fluorochromes. Each of these labels a special cell structure. The fluorescence images will then be acquired. The fluorescence image 1, created with fluorochrome 1 shows cell structure 1, the fluorescence image 2 created with fluorochrome 2 shows cell structure 2, etc.. The individual images will be combined into a multi-channel fluorescence image that shows the different cell structures in different colors. When, for example, three fluorochromes are used, a three channel fluorescence image will be created.



Problem with the visual separation of the structures

Filter sets in the microscope

Your microscope has an appropriate filter set for each fluorochrome, this set comprises an excitation filter, a dichromatic mirror, and an emission filter. When the fluorochrome 1 is excited by light from the wavelength range 1a, it emits light in the wavelength range 1b. When fluorescence image 1 is acquired, the excitation filter 1 takes care that only light from a narrow range within the wavelength range 1a reaches the sample from the microscope's illumination source. At the same time, the dichromatic mirror 1 and the emission filter 1 take care that, from the sample, only light from a narrow range within the emission wavelength range 1b, reaches the camera.

Overlapping of the wavelength ranges

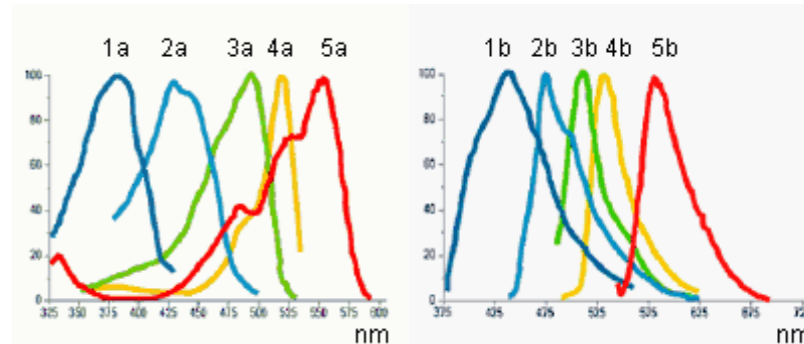
The problem with this procedure is, that the wavelength ranges of the different fluorochromes overlap. If this overlapping didn't occur, the aspired visual separation of the different cell structures in the resulting multi-channel fluorescence image would be perfect.

Neither the excitation wavelength ranges nor the emission wavelength ranges have sharp limits, and they lie very close to one another, where numerous fluorochromes are concerned. Therefore, the excitation wavelength ranges 1a, 2a, 3a, ... of the fluorochromes 1, 2, 3, ... normally overlap. The same applies to the emission wavelength ranges 1b, 2b, 3b ... As well as that, there are also

overlappings between excitation wavelength ranges and emission wavelength ranges.

Excitation and Emissions spectra

In the spectra that follow, you can see a graphical demonstration of the way in which the excitation intensities and the emission intensities of several fluorochromes that are often used, depend on the wavelength. The way in which the different wavelength ranges overlap, can clearly be seen in these spectra.



Spectral unmixing

Owing to the spectral overlapping, the aspired visual separation of the different cell structures only succeeds partially. When, for example, the light that excitation filter 1 lets through, also excites fluorochrome 2 a little, and part of the light that fluorochrome 2 then emits can pass the emission filter 1; cell structure 2 will also be dimly visible in fluorescence image 1. One can then speak of an unwanted "spectral mixing" of the individual fluorescence images.

Spectral unmixing

The spectral mixing can be subsequently removed from a digitally recorded multi-channel fluorescence image, by recalculation. That's to say, the image will be "spectrally unmixed". When you do this, it improves the visual separation of the different cell structures in the image, and improves the image quality. To do that, use the [Process > Enhancements > Fluorescence Unmixing...](#) command.

00360

10.2.1. Carrying out a fluorescence unmixing

The spectral unmixing of a multi-channel fluorescence image takes place in two steps. The first step is the calibration of the color channels with the help of reference images. When the experimental conditions don't change, you will only need to carry out this step once. In the second step, the actual spectral unmixing takes place.

You require precisely one reference image for each color channel that is to be calibrated. Each reference image must have exactly the same number of color channels as the image that is to be unmixed. In the instructions that follow, it will be assumed that you have acquired a three channel fluorescence image and want to carry out a spectral unmixing with it. For a two channel fluorescence image the procedure is analogical.

Calibrating color channels

When the experimental conditions don't change, you will only need to carry out the calibration once. When you've done that, you can spectrally unmix all of the three channel fluorescence images that are acquired later, on the basis of this calibration.

Acquiring reference images

1. Set up three samples that in each case have only been stained with one of the three fluorochromes.
Alternatively, you can use a single sample that has been stained with all

three fluorochromes. In this case, there must be three areas on the sample that have each been stained with only one fluorochrome.

2. Acquire a three channel fluorescence image of each of the three samples (alternatively, of each of the three areas on your sample).
When you do this, use either the excitation filter appropriate for each of them, and a multiband emission filter or a multiband excitation filter and the emissions filter appropriate for each of them.
The experimental conditions must be the same as they were when the image that is to be spectrally unmixed was acquired.
 - Differences in the exposure times of the individual color channels will be automatically linearly corrected when a spectral unmixing is carried out. Nevertheless, as a rule it makes sense not to change the exposure times when the reference images are acquired.
 - The result will be three multi-channel fluorescence images. Each of them contains three channels. These will, in what follows, be designated as "reference image 1", "reference image 2" and "reference image 3". Reference image 1 is to be used to calibrate color channel 1, that belongs to fluorochrome 1. With the reference images 2 and 3 the method is analogical.
 - Each of the reference images will be displayed in its own window, in the document group. The reference image that was last acquired will be the currently displayed, active image.
 - Should you have already acquired the three reference images at an earlier point in time, you can load them into the document group by using the *File > Open > Image...* command.

Defining ROIs



1. Activate reference image 1 in the document group.
2. In the *Life Science Applications* toolbar, click the *New ROI - 3 Points Circle* button.
 - Should the toolbar not be visible, put it on display by using the *View > Toolbars > Life Science Applications* command.
3. Search out an area in reference image 1, in which fluorochrome 1 is especially bright and glows as evenly as possible.
4. Define a circular ROI within this area with three mouse clicks.
 - This ROI was defined for the fluorochrome 1. It will be automatically assigned the name "ROI 1".
 - You can still subsequently change the size and position of this ROI.
 - You can change this automatically created name. To do so, use the *Measurement and ROI* tool window. In it, click the ROI's name to change it. Should the tool window not be visible, put it on display by using the *View > Tool Windows > Measurement and ROI* command.



5. Click the *New ROI - 3 Points Circle* button once more.
6. Search out a dark area in the background of reference image 1, in which, as far as possible, no fluorochrome can be seen.
7. Define a circular ROI within this area with three mouse clicks.
 - This ROI was defined for the image background. It will be automatically assigned the name "ROI 2".
8. Using the same procedure, define in reference image 2 an ROI for the fluorochrome 2, and an ROI for the image background.
9. Using the same procedure, define in reference image 3 an ROI for the fluorochrome 3, and an ROI for the image background.

Finishing the calibration

1. Activate reference image 1 in the document group.
2. In the *Life Science Applications* toolbar, click the *Fluorescence Unmixing* button to open the *Fluorescence Unmixing* dialog box.
3. Activate the *Calibration* tab.
4. Enter the label for fluorochrome 1 in the *Name* field. This is, at the same time, the name for the calibration for color channel 1.
5. Select reference image 1 in the *Image* list.
6. In the *ROI* list, located immediately below the *Image* list, select "ROI 1" which was defined for the fluorochrome 1.
7. In the *Background subtraction* group, select the *ROI* option for the background correction of reference image 1.
8. In the neighboring list to the right, select "ROI 2" that was defined for the image background in reference image 1.
9. Click the *Save* button.
 - The calibration of color channel 1, has now been completed.
 - The *Fluorochrome 2 >>* button will become available.
10. Click the *Fluorochrome 2 >>* button to skip to the calibration of color channel 2.
 - The name of the *Fluorochrome 1* group will then change to *Fluorochrome 2*.
11. Then, using the same procedure, calibrate color channel 2 with reference image 2 and fluorochrome 2.
12. Then, using the same procedure, calibrate color channel 3 with reference image 3 and fluorochrome 3.
13. Click the *Cancel* button, to close the *Fluorescence Unmixing* dialog box.

Spectrally unmixing a three channel fluorescence image

1. In the document group, activate the three channel fluorescence image you want to spectrally unmix.
 - Should you have already acquired the image at an earlier point in time, you can load it into the document group by using the *File > Open > Image...* command.



2. In the *Life Science Applications* toolbar, click the *New ROI - 3 Points Circle* button.
3. Search out a dark area in the background of your image, in which, as far as possible, no fluorochrome can be seen.
4. Define a circular ROI within this area with three mouse clicks.
 - This ROI was defined for the image background. It will be automatically assigned the name "ROI 1".



5. In the *Life Science Applications* toolbar, click the *Fluorescence Unmixing* button to open the *Fluorescence Unmixing* dialog box.
6. Activate the *Linear Unmixing* tab.
7. In the *Fluorochrome 1* list, select the calibration with which the fluorescence image in your multi-channel fluorescence image's color channel 1 is to be corrected.
 - The name of this calibration is identical with the label you gave the fluorochrome 1 while you were performing the calibration.

8. In the *Fluorochrome 2* list, select the calibration with which the fluorescence image in your multi-channel fluorescence image's color channel 2 is to be corrected.
9. In the *Fluorochrome 3* list, select the calibration with which the fluorescence image in your multi-channel fluorescence image's color channel 3 is to be corrected.
10. In the *Background subtraction* group, select the *ROI* option for the background correction of your image .
11. In the neighboring list to the right, select "ROI 1" which was defined in your image for the image background.
12. Click the *OK* button to carry out the spectral unmixing and to close the dialog box.
 - A new image document will be created for the spectrally unmixed image. The source image will not be changed.
 - It can occur that, immediately after the spectral unmixing, the image will not be optimally displayed on your monitor. In this case, click the *Apply* button in the *Adjust Display* tool window. When you do this, the image contrast on your monitor will be automatically optimized. The actual image data will not be changed.
13. Save the spectrally unmixed image if you need it.

00361

10.3. Colocalization

10.3.1. Overview

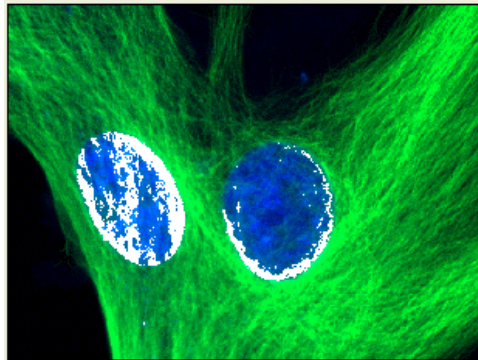
What is colocalization?

In the fluorescence microscopy, it can occur that the fluorescence signals emitted by two parts of a sample (e.g., molecules) that have been stained with different fluorochromes, interfere with each other. In these cases, the different parts of the sample lie very close to one another, or one over the other. The effect of the interference of fluorescence signals is termed "colocalization".

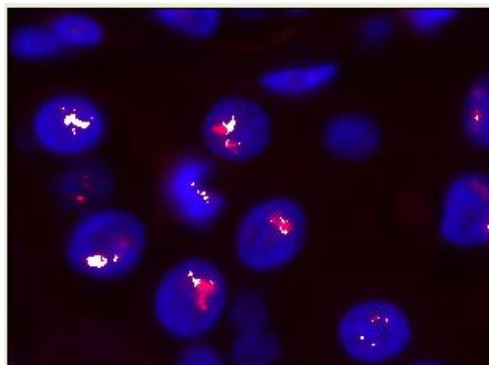
In the digital image analysis, the colocalization of fluorescence signals can be measured. This is done by detecting pixels that have the same intensity in both color channels. These measurements are carried out on multi-channel images, and are always valid for one channel pair.

Examples for colocalization

1) Superimposed signals in the green and blue color channels. The colocalized pixels are displayed in white



2) Superimposed signals in the blue and red color channels. The colocalized pixels are displayed in white



00705 26072011

10.3.2. Measuring the colocalization

Use the *Colocalization* button to start a measurement of colocalization. You will find this button on the *Life Science Applications* toolbar. This button isn't available in all software versions.

Measuring the colocalization on the whole frame



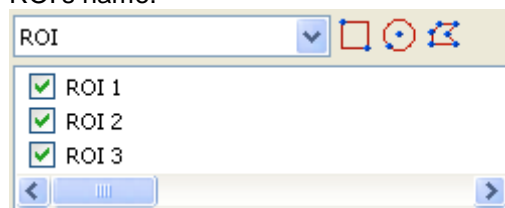
1. Load the multi-channel image you want to use for the colocalization measurement.
2. On the *Life Science Applications* toolbar, click the *Colocalization* button. If this toolbar is not displayed, use the *View > Toolbars > Live Science Applications* command.
 - The *Colocalization* dialog box opens.
3. In the *Channels* field, choose the two color channels for which the measurement of colocalization is to be carried out.
4. When you work with multi-channel time stacks or multi-channel Z-stacks: Determine in the *Apply on* group, whether the colocalization measurement is to be carried out on all frames or only on selected frames. Should you want to limit the image selection, select the *Selected frames* entry, then click the *Dimension Selector* button.
 - Then you can limit the image selection in the *Dimension Selector* tool window. You can find more information on this tool window in the online help.
5. In the *Target area* group, select the *Whole frame* entry, in the *Area* field.
6. Click the *Options...* button, then select the *Colocalization channel (Image)* and *Measurement results (Workbook)* check boxes.
7. Pay attention to the displayed results in the preview, and in the *Results* group.
8. If necessary, change the position and size of the intensity range in the scatterplot.
 - Then only the colocalization of the pixels that lie within the chosen intensity range are shown in the preview.
9. Click the *OK* button to finish the measurement of colocalization.
 - A new image that contains the colocalization channel will be created.
 - At the same time, a workbook that contains the results of the colocalization measurement, will be displayed.
10. If required, use the *File > Save as...* menu command, to save the new image and the workbook.

Measuring the colocalization on a part of the image (ROI)

Frequently, a colocalization of fluorescence signals occurs only in a small image segment. In this case, it makes sense to define a ROI (Region of Interest) then determine the colocalization only within this ROI. You can also define several ROIs. ROIs can have any shape you wish. You can find general information on setting up a template in the online help.



1. Load the multi-channel image you want to use for the colocalization measurement.
2. On the *Life Science Applications* toolbar, click the *Colocalization* button. If this toolbar is not displayed, use the *View > Toolbars > Live Science Applications* command.
 - The *Colocalization* dialog box opens.
3. In the *Channels* field, choose the two color channels for which the measurement of colocalization is to be carried out.
4. When you work with multi-channel time stacks or multi-channel Z-stacks: Determine in the *Apply on* group, whether the colocalization measurement is to be carried out on all frames or only on selected frames. Should you want to limit the image selection, select the *Selected frames* entry, then click the *Dimension Selector* button.
 - Then you can limit the image selection in the *Dimension Selector* tool window. You can find more information on this tool window in the online help.
5. Click the *Options...* button, then select the *Colocalization channel (Image)* and *Measurement results (Workbook)* check boxes.
6. In the *Target area* group, click once in the *Area* field, to open the picklist. Select the ROI entry.
 - Next to the field, to the right, the buttons with the various ROI forms are displayed.
7. Click the button for the required ROI form that you want to set up. You have the choice between a rectangle, a circle and a polygon.
 - The mouse pointer will appear in the image window. The *Colocalization* dialog box is hidden.
8. Define the first ROI with clicks of your left mouse button. When you have completed the definition of your ROI, click your right mouse button, then select the *Confirm Input* command in the context menu.
 - You will then once more see the *Colocalization* dialog box. The ROI you have defined will now be shown in the preview image.
9. If required, define further ROIs.
10. Select the required ROIs. To do this, click once in the box to the left of the ROI's name.



11. Pay attention to the displayed results in the preview, and in the *Results* group.

12. If necessary, change the position and size of the intensity range in the scatterplot.
 - Then only the colocalization of the pixels that lie within the chosen intensity range are shown in the preview.
13. Click the *OK* button to finish the measurement of colocalization.
 - If you haven't changed the default settings for colocalization, a new image, that contains the colocalization channel, will be created.
 - At the same time, a workbook that contains the results of the colocalization measurement, will be displayed. The columns in the workbook contain the supplement "ROI".
14. If required, use the *File > Save as...* menu command, to save the new image and the workbook.
15. The multi-channel image will also have been changed when the ROI was defined. Therefore, if you want to keep the ROI, save it also.

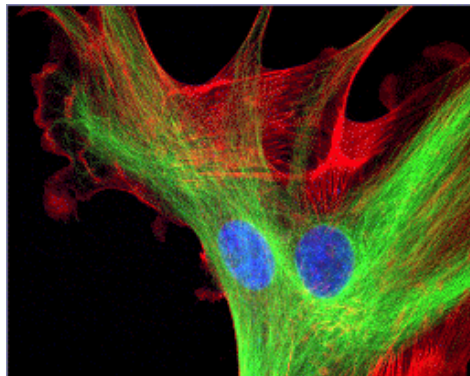
Measuring the colocalization on a color channel


You can also measure the colocalization on image structures. Where images are concerned on which the image structures that are to be analyzed are numerous, and are spread over the whole image, this procedure is quicker than setting a lot of ROIs. If, for example, you want to measure the colocalization on image structures that have been stained with the (blue fluorescent) fluorochrome DAPI, select the blue channel. Then define the threshold values for this channel.

The colocalization measurement on a color channel only makes sense on a multi-channel image with at least three color channels.

Example:

On the image, the colocalization of the red and green pixels within the area marked in blue (cell nucleus) is to be measured. All other positions on the image where pixels colocalize are to be ignored.

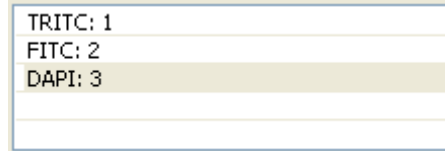


Measuring the colocalization on a color channel 


1. Load the multi-channel image for which you want to carry out a colocalization measurement.
2. On the *Life Science Applications* toolbar, click the *Colocalization* button.
 - The *Colocalization* dialog box opens.
3. When you work with multi-channel time stacks or multi-channel Z-stacks: Determine in the *Apply on* group, whether the colocalization measurement is to be carried out on all frames or only on selected frames. Should you want to limit the image selection, select the *Selected frames* entry, then click the *Dimension Selector* button.
 - Then you can limit the image selection in the *Dimension Selector* tool window. You can find more information on this tool window in the online help.

4. Click the *Options...* button, then select the *Colocalization channel (Image)* and *Measurement results (Workbook)* check boxes.
5. In the *Target area* group, select the *Channel segmentation* entry, in the *Area* field.

- In the *Area* field, a picklist with all of the available channels will open.



6. Select the channel of the fluorochrome with which the image structure that is to be analyzed, has been stained. In the example shown above, this is the "DAPI" channel.

7. Click the button  located next to the *Area* field, on its right-hand side.

- A picklist with the different methods for setting threshold values, will open.

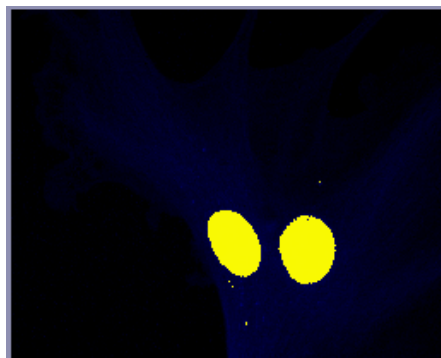
8. Select the *Automatic Threshold...* method.

- This method requires the user to make the smallest number of settings. Therefore, you should only use the other methods for setting threshold values, when the *Automatic Threshold...* method doesn't lead to the result you wanted. You can find an overview on the subject of threshold values in the online help.
- The *Separation Channel Threshold* dialog box opens. Your software will carry out an automatic setting of threshold values. In the image window, you will now see the image structures that are detected by the automatic threshold settings.

9. In the *Channel* group, select the required channel again (in this case "DAPI").

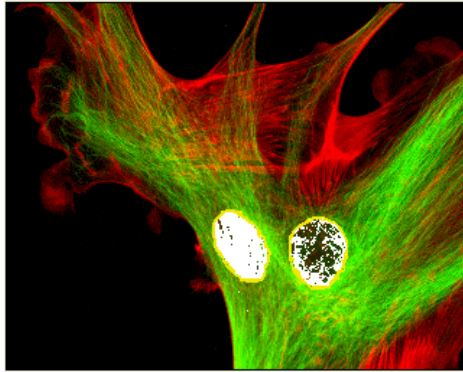
10. Check in the image window, whether the automatic threshold setting has correctly found the image structures that are to be analyzed.

- In the *Separation Channel Threshold* dialog box, select the *Dark* or *Bright* option in the *Background* group, should the *Automatic* option not lead to the results you want.



11. When the image structure that is to be analyzed has been correctly found, click the *OK* button.

- You will then once more see the *Colocalization* dialog box. In the preview, the image structures found via the Channel segmentation will now be displayed with a yellow outline. The colocalized pixels shown lie exclusively within these image structures.

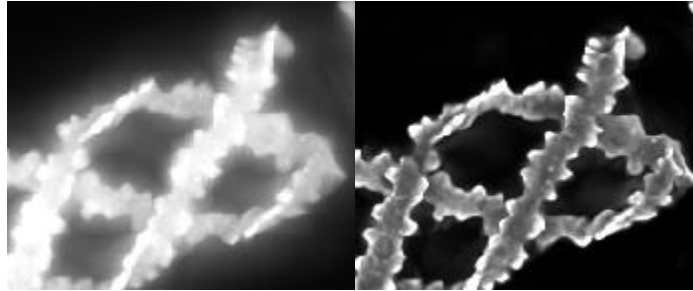


12. If required, change the position of the white rectangle (gate) in the scatterplot. By doing this you'll change the observed intensity range. You can now, e.g., have pixels with a lower colocalization shown. Pay attention to the display in the *Results* group.
13. Click the *OK* button to finish the measurement of colocalization.
 - A new image that contains the colocalization channel will be created.
 - At the same time, a workbook that contains the results of the colocalization measurement, will be displayed. The columns in the workbook contain the supplement "Separation channel".
14. If required, use the *File > Save as...* menu command, to save the new image and the workbook.
15. The multi-channel image will also have been changed when the Channel segmentation was defined. Therefore, if you want to keep these settings, save it also.

00709 26072011

10.4. Deconvolution

The *Process > Deconvolution* submenu offers you deconvolution filters with which you can remove disturbing diffused light from an individual image or a multi-dimensional image. With a suitable parameter selection the image will become sharper and more clear.



Before using a deconvolution filter

The result of a deconvolution process largely depends upon whether certain parameters are known with which the image was acquired. These parameters include for example the objective's numerical aperture and refraction index. Before using a deconvolution filter on an image, use the *Process > Deconvolution > Verify Channel Parameters...* command, to check the relevant parameters for the image, changing them if necessary. A description of this dialog box can be found in the online help.

What is deconvolution?

In fluorescence and brightfield microscopy, diffused light from areas above or below the focal plane leads to over exposure, distortion and blurring. A suitable mathematical model to describe this problem is a convolution operation:

$$g(x) = f(x) * h(x) + n(x)$$

x: Point in XY space

g(x): Observed image

f(x): Ideal image

h(x): Point spread function

n(x): Noise function

*: Convolution

To be able to reconstruct the ideal image $f(x)$ from the observed image $g(x)$, you must know the noise function $n(x)$ and the point spread function $h(x)$. While an estimation of the noise function $n(x)$ is highly possible, the point spread function $h(x)$ depends normally so strongly on the optical properties of the microscope and the sample, that an experimental determining of this function is not directly possible. For this reason, mathematical algorithms become necessary to even approximately determine the point spread function $h(x)$ and to subsequently make the best possible reconstruction of the ideal image $f(x)$ by means of deconvolution. A perfect, unambiguous, reconstruction is generally not possible, since information can be lost during a convolution.

The deconvolution filters

The individual deconvolution filters essentially differ in how the point spread and noise functions are determined, which are needed for the deconvolution of the image and the noise depression.

The more image data is used to calculate the point spread function, the more precise the result will be and the longer the calculation will take.

2D deconvolution

The 2D deconvolution filter uses a theoretical point spread function, in which only the acquisition parameters are used but no image data.

You can apply the 2D deconvolution filter on all supported image types.

However, the 2D deconvolution filter always affects only an individual frame.

As the amount of data processed is quite small, the 2D deconvolution filter is very quick. It makes the image appear much sharper but does not then permit any quantitative analysis of the image data.

The filter is especially well suited for TIRF images where the image information comes from a very narrow Z-range of the sample.

Nearest Neighbor Filter

The nearest neighbor filter employs a theoretical point spread function for the deconvolution, in the calculation of which, the data of the image under examination and of the two neighboring images of a Z-stack, are taken into consideration. The point spread function is applied to the neighboring images.

The scaled sum of the neighboring images that have been processed in this way, is then deducted from the image under examination. With single images and simple time stacks the nearest neighbor filter works like a no neighbor filter. In this case, only the observed image's data are used in the calculation of the point spread function.

Wiener Filter

The Wiener filter approximates the point spread function by a linear function, with the mean square deviation being minimized. The actual filter is calculated from the linear inverse function. With Z-stacks the complete Z-stack's data are used in the calculation, with individual images only the observed image's data.

Constrained Iterative Filter

The **Constrained Iterative** filter does not make any presumptions about the point spread function, but rather extracts it directly from the Z-stack. This occurs iteratively. An estimated point spread function is used as a starting point. Then, an assumption is made as to which ideal image, via this point spread function, would have led to the observed image. Then, an estimation has to be made as to which point spread function caused the original image to be transformed into the observed image. This alternating assessment can be repeated as often as wished. Special mathematical processes are used to ensure that these iterations converge to reasonable values.

The **Constrained Iterative** filter promises the best results of all of the deconvolution filters, requires though, the most calculation time. Since the filter works iteratively on the complete Z-stack, a use on individual images or simple time stacks, is not possible.

00355 091032012

10.5. Ratio Analysis

10.5.1. Overview

Certain multichannel fluorescence microscopy inspection modes allow you to monitor changes in ion concentration or pH value within cellular structures. Fluorescence dyes whose excitation characteristics depend on the concentration of ions are used for this.

The Fura-2 fluorescence dye, for example, shifts its excitation level from 340 nm to 380 nm when the calcium ion concentration decreases. At an excitation wavelength of 340nm, the intensity increases when the calcium concentration increases. At an excitation wavelength of 380nm it's the exact opposite. The higher the calcium concentration is, the less light is emitted.

The process flow of a ratio analysis on a multi-channel fluorescence image

1. Acquiring a multi-channel fluorescence image

The fluorescence dye is excited with two different wavelengths, one after the other. The multi-channel fluorescence image contains two color channels created with the same fluorescence dye, but at different excitation wavelengths. The excitation wavelengths 340 nm and 380 nm are typically used with the Fura-2 fluorescence dye.

2. Carrying out background correction

A background correction is carried out on both color channels. You can make settings for the background correction in the *Ratio Analysis* dialog box. A description of this dialog box can be found in the online help.

3. Calculating a Ratio Image

One color channel is divided by the other on a pixel by pixel basis. The result is the ratio image, in which the intensity is proportional to the ion concentration.

When the Fura-2 fluorescence dye is used, the image acquired at the excitation wavelength of 340 nm is divided by the image acquired at the excitation wavelength of 380 nm:

$$\text{color channel (340 nm) / color channel (380 nm)}$$

4. Viewing a ratio image

The result is a multi-layer image. One image layer is the source image and the other image layer is the ratio image. The ratio image displays the concentration of ions using pseudo colors. The pseudo color image is superimposed on the source image so that you can see the structures in your sample and the concentration of ions at the same time.

You can change the display of the resulting image in the image window, in order to view only the ratio image for example.

The process flow of a ratio analysis on a multi-channel time stack

1. Acquiring a multi-channel time stack

2. Carrying out background correction

3. Calculating an intensity profile

When the ratio analysis is carried out on a multi-channel time stack, you can compute the intensity profile in addition to the ratio image. The intensity profile displays the change in concentration of ions in a particular image segment. You determine the image segment by defining a ROI. You can define the ROI right here in the *Ratio Analysis* dialog box. A description of this dialog box can be found in the online help.

You can measure the intensity profile of several image segments at the same time.

4. Viewing a ratio image

The result is a multi-layer image. One image layer is the source image and the other image layer is the ratio image. The ratio image displays the concentration of calcium ions using pseudo colors. The pseudo color image is superimposed on the source image so that you can see the structures in your sample and the concentration of calcium ions at the same time.

You can change the display of the resulting image in the image window, in order to view only the ratio image for example.

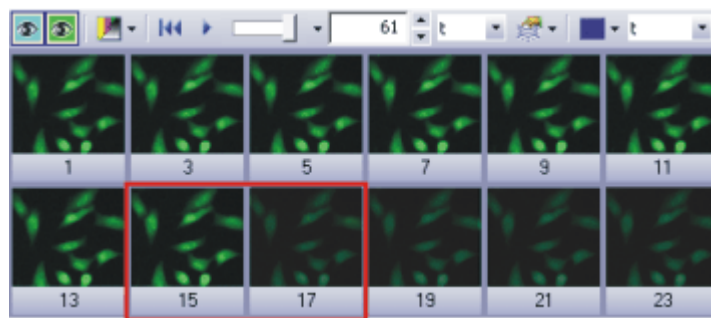
00244 25072013

10.5.2. Carrying out a Ratio Analysis

Use the *Measure > Ratio Analysis...* command to measure the concentration of calcium ions in a time stack. This command is also available as a button on the *Life Science Applications* toolbar.

Measuring changes in the concentration of calcium ions in a time stack

Task The Fura-2 fluorescence dye makes it possible to measure the concentration of free calcium ions because its excitation level shifts from 340 nm to 380 nm as the calcium ion concentration decreases. Use the ratio analysis to compute the ratio image over time and the intensity profile in two cells.



The image displays an overview of the frames in a multi-channel time stack that has 2 color channels. The sample has been dyed with the Fura-2 fluorescence dye. Between the frames that are framed in red in the illustration, the image intensity decreases visibly. The cause is a change in the calcium ion concentration.

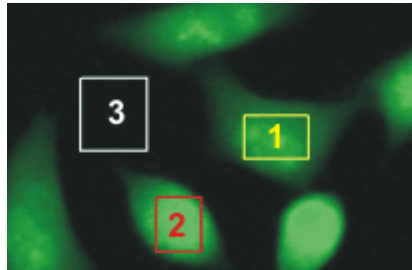
Preparing the analysis

1. Several example images were supplied together with your software. You can follow these step-by-step instructions using the Fura.tif example image. This example image is a multi-channel time stack image.
2. Use the *View > Toolbars > Life Science Application* command, to have the *Life Science Application* toolbar displayed. You can find the functions for defining ROIs and for Ratio Analysis on this toolbar.

Defining ROIs (Region Of Interest)



3. Click the *New ROI – Polygon* button on the *Life Science Applications* toolbar.
4. Draw a rectangle inside a cell.
5. Define another ROI in a different cell.
6. Define another ROI in a dark image segment that has no fluorescing objects. This ROI will be used as a reference for the background correction.
7. Rename the ROIs you defined.
To do this, open the *Measurement and ROI* tool window. In the *Measurement and ROI* tool window, double-click on the first ROI's name. Enter a descriptive name for the ROI. Name the ROIs *Cell01*, *Cell02* and *Background*, for example.

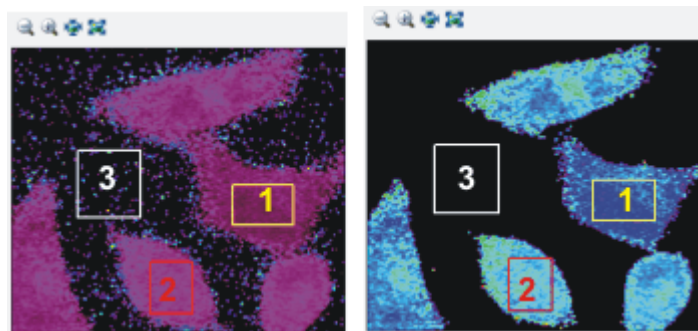


Three ROIs have been defined on the image. The red and yellow ROIs contain cells. The white ROI is on the background.

Carrying out a Ratio Analysis



8. Click the *Ratio Analysis* button located on the *Life Science Applications* toolbar.
 - The *Ratio Analysis* dialog box opens.
9. Make the settings for the background correction in the *Background* group. Select the *ROI* option. In both lists, select the reference ROI for the background correction.
10. In the *Ratio* group, select the parameters for the calculation of the ratio image. Select the *Fura340* color channel from the *Numerator* list and the *Fura380* color channel from the *Denominator* list.
 - The preview image in the *Ratio Analysis* dialog box displays the ratio image that has been computed for the time point that is currently displayed in the image window. The ratio image is the result of dividing the intensity of the Fura340 color channel by the intensity of the Fura380 color channel.
 - The ratio image is a gray-value image which automatically has a predefined pseudo color table applied to it in the preview window. High ratio values are displayed in red with this pseudo color table and low ratio values are displayed in magenta.
 - The ratio image has single pixels with a high intensity in the background. This is image noise.
11. In the *Thresholds* fields, increase the value until the image noise disappears and only the cells remain visible.
12. In the *Scale* list, accept the value of 1000 that is given.

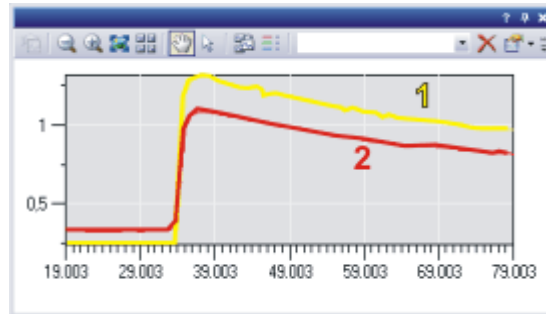


On the **left** you can see the preview image before the threshold values were set. On the **right** you can see the preview image after the threshold values were set. The image background is now black. The colors in the preview image have changed because the image in the preview window is always displayed with the most possible contrast.

Outputting results

13. In the *Output* group, select the *Image as new layer* and *Intensity Profile* check boxes. If you want to output the intensity profile as a sheet, select the *Export to workbook* check box.

14. Select the ROIs that you defined on the cells. The ROIs you have selected are highlighted in the dialog box.
15. Close the *Ratio Analysis* dialog box with *OK*.
 - If it wasn't already displayed, the *Intensity Profile* tool window is displayed automatically now. The tool window contains two intensity profiles, one for each ROI you defined.



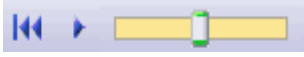
The intensity profiles show how the ratio value in both ROIs (1) and (2) changes over time. The colors of the intensity profiles correspond to the colors of ROIs they describe.

- The source image is now a multi-layer image and displays the concentration of calcium ions in addition to the image information.
16. Use the *File > Save As...* command, to save the resulting image. Save the resulting image in the TIF or VSI file format.

Setting the display of the ratio image.

1. Carry out a ratio analysis.
 - The image resulting from a ratio analysis is a multi-layer image. One image layer is the source image and the other image layer is the ratio image. There are several ways of displaying the resulting image on the monitor.
2. Use the *View > Tool Windows > Layers* command to make the *Layers* tool window appear. You have access to the individual image layers in the *Layers* tool window.
3. Select the *View > Tool Windows > Adjust Display* command to make the *Adjust Display* tool window appear. In the *Adjust Display* tool window, you can specify how an image is displayed on the monitor.
4. Activate the image resulting from the ratio analysis in the document group.

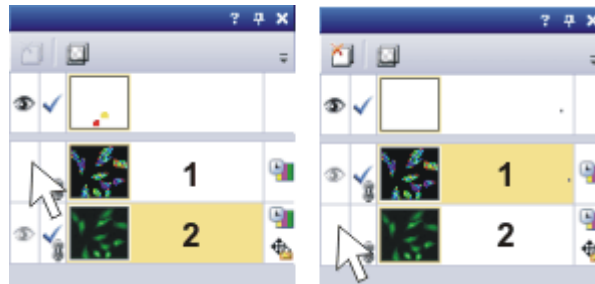
Browsing the time stack

1. Take a look at the peak in the intensity profile in this time stack. To do this, use the navigation bar at the top of the image window.
 
2. If the info stamp isn't displayed in the image window, use the *View > Info Stamp* command to display it.
 - The info stamp should display the time for each frame.
 - If this time is not on display, select the *Tools > Options...* command. In the tree view, select the *Info Stamp > Properties* entry. In the *Available properties* list, select the *Image > t* check box. Close the dialog box with *OK*.

Viewing the ratio image
and the source image
separately

You can view only the ratio image or only the source image in the image window whenever you want.

1. In the *Layers* tool window, click once on the source image to select this image layer. The name of the image layer in the tool window corresponds to the name of the image.
2. Click once on the eye icon next to the ratio image.
 - The ratio image is now not displayed in the image window. You see only the source image.
3. In the *Layers* tool window, click once on the ratio image to select this image layer.
 - When you select an image layer in the *Layers* tool window, this image layer is automatically displayed.
4. Click once on the eye icon next to the source image.
 - The source image is now not displayed in the image window. Now you see only the ratio image.



The illustration shows the *Layers* tool window with the image resulting from a ratio analysis.

On the left, the ratio image (1) is not displayed. On the right, the source image (2) is not displayed.

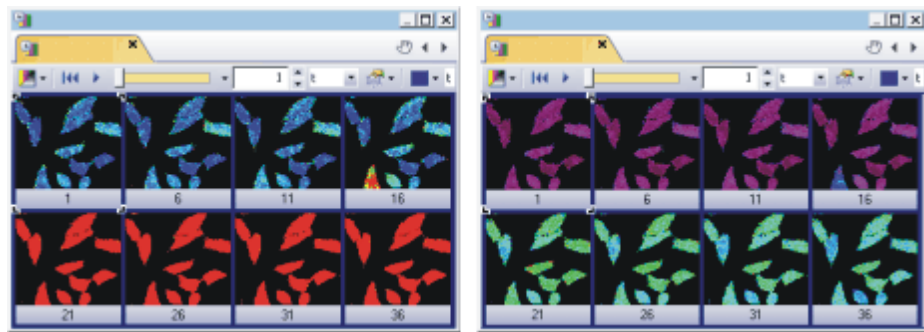
Optimize the display of
the ratio image

1. Display only the ratio image in the image window.
2. Use the *Adjust Display* tool window to optimize the display of the ratio image.
3. Select the *Auto Contrast* option.

This makes sure that the ratio image is displayed in the image window with the most possible contrast. With this setting, all the colors in the pseudo color table you are using are applied to the ratio image.

The *Histogram of all frames* check box decides whether only the frame in the time stack that is currently on display will have its contrast optimized or whether the contrast will be optimized across all the frames in the time stack.
4. Select the *Histogram of all frames* check box.

Your software now takes the smallest and largest values in all the frames and assigns the colors black and red to these values.
5. Click the *Apply* button to make the changed settings visible in the image window.



The illustration shows the same ratio image where different settings have been made in the *Adjust Display* tool window.

On the **left**, the contrast has been optimized for the first frame. The *Histogram of all frames* check box is not checked. Because the values in the ratio image increase over time, the color shifts towards red.

On the **right**, the contrast was optimized across all frames. The *Histogram of all frames* check box was selected. With this setting, differences in the ratio image can be seen in all frames.

Viewing the ratio image
and the source image
at the same time

1. Display all the image layers in the image window. In the *Layers* tool window, you can see an eye icon next to each image layer.
2. Select the ratio image in the *Layers* tool window, and click your right mouse button to open a context menu.
3. Select the *Mode > Intensity Modulation* command from the context menu. If this mode is already set, keep it.
 - The colors in the ratio image remain unchanged and their color value reflects their ratio value.
 - The intensity of the ratio image is adjusted to the intensity of the source image. Where there is a low intensity in the source image, the ratio image is also dark.

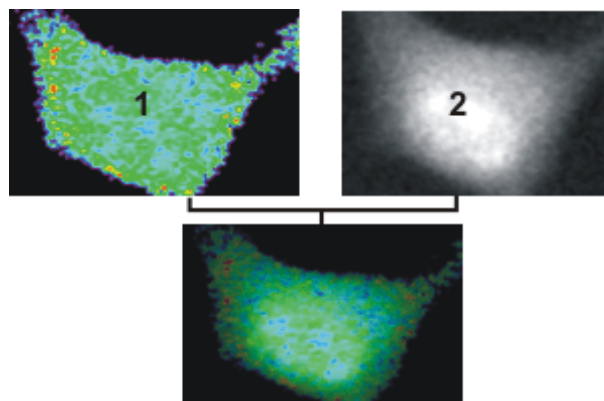


Image (1) is only the ratio image. The colors are all equally bright. Image (2) is the source image. Your software hasn't applied color mapping to it.

In the image at the bottom, the intensity in the ratio image corresponds to the intensity in the source image. The intensity decreases noticeably towards the edges of the cell.

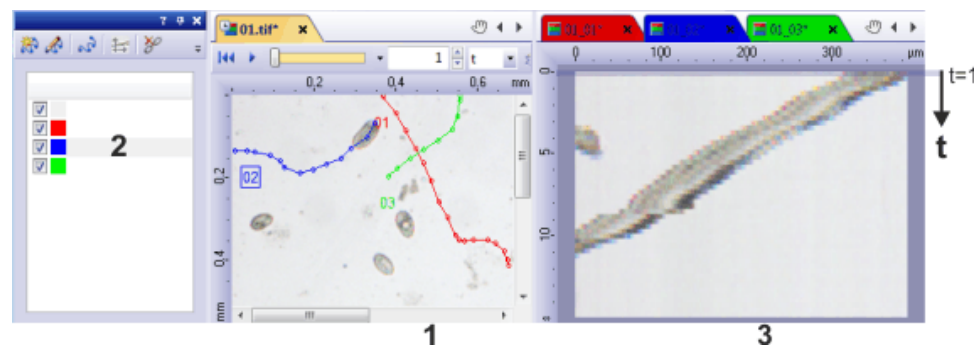
00245 25072013

10.6. Kymograph

What does the kymograph measure?

Use the *Kymograph* tool window to create a visual representation of the movement of objects in an image series.

Define one or more tracks on an image series. A track is a line that can follow any course you want. You can assign a particular width to it. For each track, the kymograph computes the intensity values along the line and plots these values against the time or the Z-value. The result is one kymograph for each defined track. The kymograph is an image that is calibrated differently on the horizontal and vertical axes. With a time stack for example, a kymograph's X-direction is calibrated in units of time and the Y-direction is calibrated in units of length.

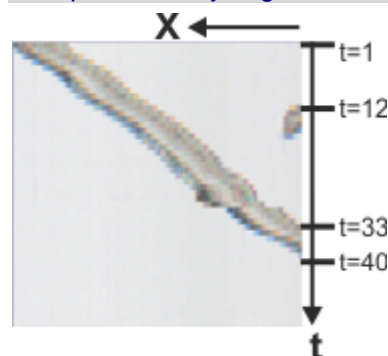


In the above example, three tracks are defined on the image series (1). The kymograph is computed for each track. The kymographs are automatically arranged to the right of the image series.

Each of the tracks has been assigned a different color. The header of the corresponding kymograph in the document group has the same color.

In the illustration, the blue track (2) is selected in the tool window. This displays the corresponding kymograph (3) in the document group to the right of the tool window. The topmost line in the kymograph shows the intensity profile along the blue line at time point $t=1$. The dark object is at the very right of the track at this time point. The kymograph shows that the object keeps moving further to the left.

Properties of kymographs



The illustration shows a kymograph that has been computed from a time stack in which dark objects move across a light background.

The intensity values along a line in the image are plotted along the image's X-direction. The line has been defined to follow an object's track precisely. The time is plotted along the Y-direction.

At time point $t=1$ the object is right at the start of the track.

At time point $t=12$ an additional object enters the image.

At time point $t=33$ the object starts to move out of the image.

After time point $t=40$ no object is visible on the defined track.

Saving the results

You can save a kymograph like a normal image. Use the VSI or TIF image format to preserve the calibration.

Note: The interactive measurement functions in the *Measurement and ROI* tool window cannot be used for the measurement of kymographs.

When you save an image series that has tracks defined on it, the tracks are saved together with the image. Use the *Kymograph* tool window to re-compute the kymographs from the tracks at the press of a button.

00551

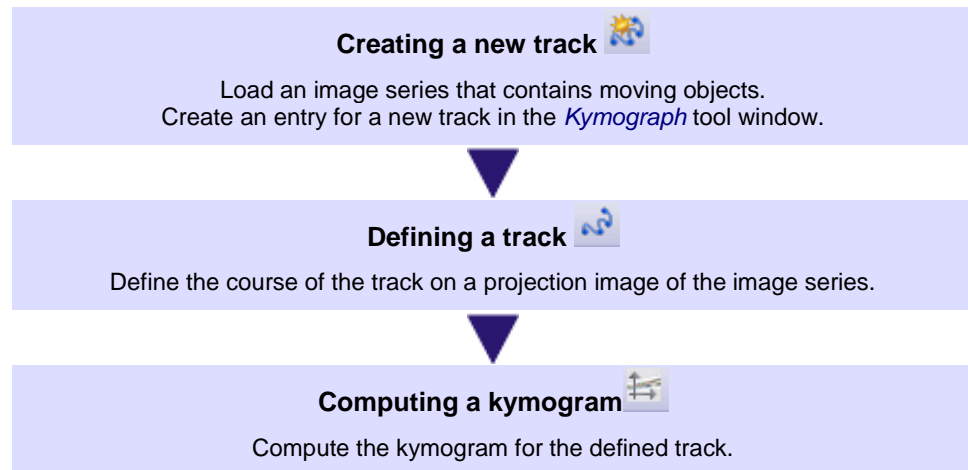
10.6.1. Working with the kymograph

Use the *Kymograph* tool window to create a visual representation of the movement of objects in an image series.

Visual representation of periodic movement

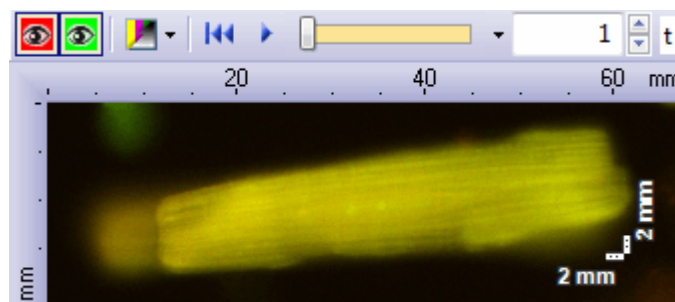
Task You have acquired a multi-channel time stack. The intensity of the fluorescence constantly rises and falls within the sample. You want to create a visual representation of the intensity profile.

The following process flow chart displays the basic steps of the process.



Preparing the analysis

1. Load the image series that you want to analyze.
 - When you load a multi-channel time stack, it will be automatically displayed in the Single Frame View in the image window.
2. When working with the kymograph you should display horizontal and vertical scale bars in the image. To do this, use the *Tools > Options...* command. Select the *Scale Bar > Display* entry in the tree view. Select the *Orientation > Horizontal and vertical* option.
3. If the scale bars aren't displayed in the image window, select the *View > Scale Bar* command to show them.



The illustration shows the first image of a multi-channel time stack.

Creating a new track

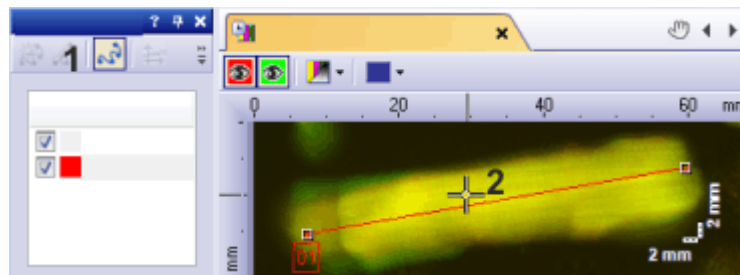


4. Use the *View > Tool Windows > Kymograph* command to make the *Kymograph* tool window appear.
5. Click the *Create Track* button in the *Kymograph* tool window to create a new track on the active image series.
 - The *Create Track* dialog box opens.
6. Enter a name for the track in the *Track definition > Name* field, *Heart_Muscle-01* for example. In the *Color* field select a color that is easy to see on the image, red for example. In the *View* list, check whether the correct projection method is being used. In this example, because the moving object is light and the background is dark, select the *Maximum intensity projection* entry from the list. Close the *Create Track* dialog box with *OK*.

Defining a track



7. Click the *Define Track Polyline* button in the *Kymograph* tool window.
 - The image series is automatically displayed in the *Maximum Intensity* projection view in the image window.
 - The mouse pointer jumps to the image window and turns into a cross.
 - All of your software's other functions are now blocked.

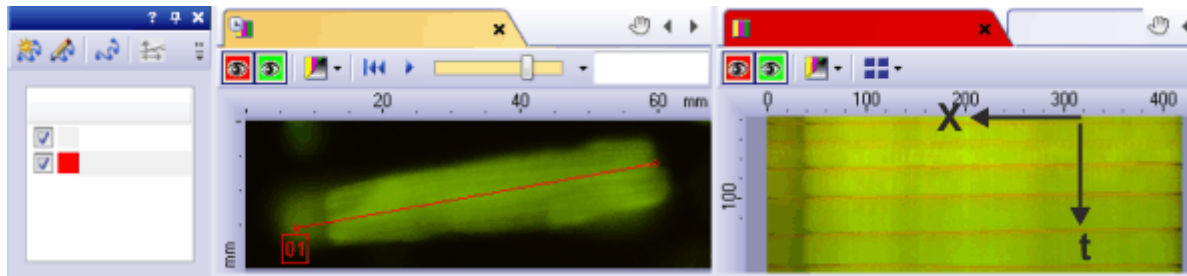


The *Define Track Polyline* (1) button appears clicked to indicate the current mode. Define the track (2) on the image series' projection view.

Computing a kymogram



8. Define the track by left clicking.
9. Right click to finish the definition of the track.
10. Click the *Compute Kymogram* button in the *Kymograph* tool window to compute the kymogram for the defined track.
 - The kymogram computes the intensity values along the course of the track and plots these values against the time. The result is a kymogram. The kymogram is a normal image that is calibrated differently on the horizontal and vertical axes.
 - The kymogram is automatically arranged to the right of the source image.
 - The kymogram's name is made up of the image's name plus the name of the track.
 - The header of the kymogram in the document group is the same color as the track.
 - The kymogram is computed separately for each color channel. Use the color channel buttons in the navigation bar to view the kymograms for the individual color channels.



On the left of the illustration, the *Kymograph* tool window is displayed. A red track is defined on it.

You can see the time stack in the center. In this case it's a multi-channel time stack. The red track runs straight across the object. This object is muscle tissue. The kymogram on the right clearly shows a periodic movement of the object. The tissue contracts and expands. A coordinate system with the image dimensions is shown on the kymogram. The width of the kymogram is defined by the length of the track. The height of the kymogram is determined by the number of frames in the time stack.

Saving the results

11. Activate the image series in the document group. Select the *File > Save As...* command and save the image series together with the defined tracks. Use the TIF or VSI image file format.
12. Activate the kymogram in the document group. Select the *File > Save as...* command and save the kymogram as an image file. Use the TIF or VSI image file format.

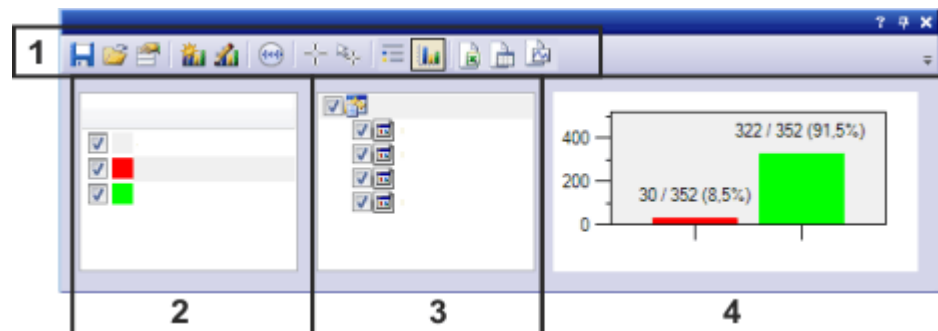
00550 10072014

11. Measuring images

11.1. Counting objects







Use the *Object Counting* tool window to manually count objects on your images. To do so, just click the objects on the images. You can define different object classes and, directly while counting, assign the objects the class you want. Use the *View > Tool Windows > Object Counting* command to have the tool window displayed.



Structure of the tool window



- (1) Toolbar of the tool window
- (2) Defining, selecting, and editing object classes
- (3) Selecting images
- (4) Viewing the results

Toolbar in the tool window

	Saving, loading, and managing class definitions	You can save your objects' class definition to a parameter set. Then, the next time you want to count objects, you can simply load the object classes and use them again. You can find more information on working with parameter sets in the online help.
	Creating class	Click this button to create a new class.
	Editing classes	Click this button to change the active class definition.
	Showing or hiding the digital reticle	Click this button to have a digital reticle displayed in the image window.
	Counting objects	Click this button to count objects.
	Editing objects	Click this button to correct a measurement.

	Switching the display of the results view	The results are displayed in the <i>Object Counting</i> tool window. Choose between a bar chart and a list view. Click the corresponding button to set the display you want.
	Exporting results	You can export the results to an MS-Excel file, a workbook or a chart.

1. Load the image you want to count objects on, or acquire an image. You can also count objects in live mode.
2. Select an object class in the *Object Counting* tool window or define one.
3. Click the *Count Objects* button to count objects.
 - When you count objects, you are in a special measurement mode. The button appears clicked, thereby showing you that the measurement mode is active. You can recognize this status by the button's colored background.
4. In measurement mode, you click on the objects in the active image that you want to count.
 - All counted objects are automatically assigned the active object class.

Note: In measurement mode, you can only count objects. In this mode, the majority of your software's other functions are not available.

To end measurement mode, release the *Count Objects* button. The measurement mode ends automatically when you define a new class or activate another image.



Editing objects

On the image, objects that have been counted are displayed with markers. You can delete or shift existing markers, for example, when you clicked the wrong object during counting.

Exporting results

You can export the results to an MS-Excel file, a workbook or a chart.

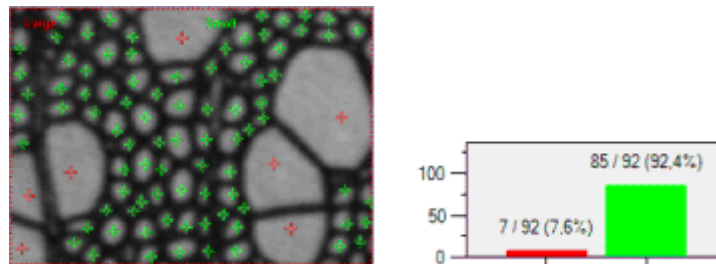
Defining, selecting, and editing object classes

Use the *Classes* area in the *Object Counting* tool window for defining, selecting, and editing object classes.

What is an object class?

While counting, you can assign the objects the class you want. For example, if you want to count small and big objects on an image, define two classes.

Note: All classes you define are only valid for the image that is currently active in the image window. If you want to use classes for several images, save the class definition and load it again for the next image.



In the image shown, small and big objects have been counted. The green class was defined for the small objects. The red class was defined for the big objects. The bar chart shows the results. 7 big and 85 small objects have been counted on the image. 92% of all counted objects are small.

Defining object classes

A class is defined by a name and a color. All objects that belong to the same class are displayed in their class color, in the image and in the bar chart. The class name is used for the labels in the bar chart and for the results sheet. The class name is also shown in the image.

There are different ways in which you can define a class.



- In the *Object Counting* tool window's toolbar, click the *Create Class* button.
- Click the *<Enter class name>* entry, in the *Classes* area.
- Right click the *Classes* area of the tool window. From the context menu, select the *Create Class* command.
- Adopt a class definition from another image. To do so, select the image that contains the class definition you want, in the *Counting Documents* group, in the *Object Counting* tool window. Then select the *Copy Class Definitions to Active Document* context menu.

Selecting images

Use the part in the middle of the *Object Counting* tool window in order to manage images on which objects have already been counted.

List of all measured images

All images on which objects have already been counted are listed below the *Counting Documents* entry, in the *Object Counting* tool window. The active image is shown in bold in the list.

Showing and hiding results

Select the check box next to the image name in order to have the results shown in the results view, in the right part of the tool window.

Clear the check box next to an image name in order to have the corresponding results hidden in the results view.

Select the check box next to the topmost *Counting Documents* entry, to have the results for all images displayed at a time. Clear the check box to hide all results at a time.

When counting objects, the results of all selected images are added up.

When using the same object classes on several images, all objects that belong to this object class are added up. When using different object classes, the results view shows all object classes that have been defined.

Note: In the *Classes* area, located on the left of the tool window, you only see the classes that have been defined for the active image. In the results view, in the right part of the tool window, all classes that have been defined on the selected images are shown.



In the example shown, objects have been counted on 4 images. On the left, only the results of the images (1) and (4) are taken into account. On the right, the results of all images are taken into account. You can see, that an additional object class has been counted on images (2) and (3).

Viewing the results

The results are displayed in the *Object Counting* tool window. Choose between a bar chart and a list.

Which results are shown?

The results show the number of counted objects per object class. Additionally, the list view shows how many objects have been counted in total.

All images that are selected in the tool window's tree view are considered for the calculation of the measurement result.

12250 14072014

11.2. Using interactive measurement functions

11.2.1. Overview

Your software offers a wide range of measurement functions. They enable you to quickly count objects and measure segments and areas. All the results will be saved together with the image and can also be issued as a sheet.

Prerequisite For making measurements, correctly calibrated images are an essential prerequisite.

Images that you have acquired with your software will have been automatically correctly calibrated when you have specified the objective you used. Should the image not yet have been calibrated, use the *Image > Calibrate Image...* command to carry out a calibration.


Additional measurement functions in your software In addition to the interactive measurement functions, your software offers you a further range of measurement functions.

Life Science Applications	The <i>Life Science Application</i> toolbar offers you various evaluation methods for your images.
Line Profile	Use the <i>Line Profile</i> tool window to measure the intensity profile along a line on an image.
Object Counting	Use the <i>Object Counting</i> tool window to manually count objects on your images.
Automatic image analysis	You can detect and analyze objects in images with your software.

Selecting the measurement environment

Measuring with help of the tool window

Switch to the *Count and Measure* layout when you want to measure images. You can find the *Measurement and ROI* tool window in the bottom section of this layout. In this tool window you have fast access to all measurement functions and settings which effect measurements. This tool window is at the same time the measurement display and contains all of the values that have been measured on the active image.

Note: Should, right at the bottom of the user interface, several tool windows lie one over the other, activate the *Measurement and ROI* tool window, by clicking on the header of the  *Measurement and ROI* tab. The tabs can be found under the tool windows.

Starting a measurement

Begin a measurement by selecting the measurement function you want. You will find the measurement function in the *Measurement and ROI* tool window, on the *Measurement and ROI* toolbar, or in the *Measure* menu.

Working in the measurement mode

As soon as you have clicked a measurement function, your software will automatically switch to a measurement mode. In the measurement mode your mouse pointer will take on the shape of a cross on the image. A small icon

indicating the selected measurement function attaches itself to the bottom right of the mouse pointer.

You can make as many measurements on the active image as you like using the measurement function that has been selected. The continuous measurement mode is valid for all loaded images. You can, therefore, easily measure numerous images one after the other.

The selected measurement function's button will keep its clicked appearance and in this way show you the current measurement function. You can recognize this status by the button's background color.

Finishing the measurement mode



You can switch off the measurement mode definitively. To do this, click on the active measurement function's button again.

You automatically turn off the measurement mode when you switch to a different mouse pointer mode. For example, click the *Select Measurement Objects* button to switch to the selection mode. You can find the button either in the *Measurement and ROI* tool window or on the toolbar. You can select and edit measurement objects in this mouse pointer mode.

Changing the default measurement mode

The continuous measurement mode described above is preset by default. You can change this default setting. To do this, use the *Tools > Options...* command. Select the *Measurement and ROI > General* entry in the tree view. Select the *Switch to 'Select Measurement Objects' mode after creating a measurement object* check box. Then, when you have completed a measurement, you will automatically leave the measurement mode again. This means you have to select the measurement function again before you start each interactive measurement.

Displaying and saving measurement results

The measurement results will be displayed directly on the image and in the *Measurement and ROI* tool window. Should this tool window not be visible, use the *View > Tool Windows > Measurement and ROI* command to bring up the tool window.


Saving the measurement results

The measurements will be saved along with the image, if you save the image in the TIF or VSI file format. You can, however, also export the measurement results in a results sheet, and save this as a file.

Showing and hiding measurement results in an image

The measurement results will be shown on the image in a special data layer, the measurement layer. On your monitor, image and measurement layer are shown together. The data of each, however, is individually stored if you use the TIF or VSI image file format. Try and picture the measurement layer as a transparency which is placed over the image. When you measure an image, the image data will not be changed by having the measurement results displayed on it.

You can, at any time, hide or show the measurement layers.

To do so, use the *Layers* tool window. There you have access to all of an image's layers. The eye icon  identifies all of the layers that are currently on display on your monitor.

Click the eye icon in front of the *Measurement and ROI* layer to hide the measurements. Click an empty cell without an eye icon to make the corresponding layer reappear.

Editing measurements

You can edit existing measurement objects at any time. The measurement values in the *Measurement and ROI* tool window will be correspondingly updated.

Note: When you load an image file with measurement objects, it is only possible to edit the measurement objects if the image file has been saved in the TIF or VSI image file format.

Selecting measurement objects



Before you can edit measurement objects, you have to select them. To do so, click the [Select Measurement Objects](#) button, and then select the measurement object(s). You can find the button either in the [Measurement and ROI](#) tool window or on the toolbar.

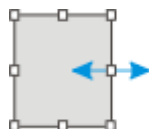
If the image is very large and many measurement objects have been defined, it can be difficult to find a particular measurement object in the image. In this case, select the measurement object that you are searching for in the [Measurement and ROI](#) tool window. Click your right mouse button and select the [Navigate to Measurement Object](#) command in the context menu. The measurement object you are looking for is then displayed in the image window.

Moving measurement objects

You can move a whole measurement object while keeping the left mouse button pressed.

Increasing/decreasing the size of measurement objects

You can also change the size of a measurement object. Move the pointer onto a marker. By dragging the marker with the mouse button depressed, you can adjust the frame's size as wished.



[Change the measurement object by moving the handles.](#)

Deleting measurement objects

Click the [Del] key on your keyboard in order to delete the selected measurement object. You can select measurement objects that you want to delete in the image and also in the sheet in the [Measurement and ROI](#) tool window.

Changing the color, font, and line thickness of individual measurement objects

You can, at any time, change the color, font, and line thickness, of individual measurement objects. Select one or more measurement objects in an image and click your right mouse button to open a context menu. In the context menu you'll find the following commands. You can use them to change the appearance of the selected measurement objects.

Measuring in the live mode

All of the measurement functions are also available in the live-image. You can therefore, e.g., quickly measure a segment in the live-image.

When you finish the live mode with the [Acquire > Snap](#) command, the measurements that you carried out in the live-image are applied to the image that was acquired.

Measuring on different image types




Measuring on image series

You can combine a series of individual images into one image. What results is e.g., a time stack in which all of the frames will have been acquired at different times.

You can make measurements on every separate image. Display the required frame on your monitor. To do this, use the navigation bar in the image window. Then carry out the measurement on this frame. The measurement will be permanently linked to this frame, i.e., the measurement will only be displayed on your monitor when the frame on which you made this measurement is also on display.

The measurement results will be shown in the [Measurement and ROI](#) tool window. You can give every measurement the number of the frame on which it was made. To do so, use, e.g., the measurement parameter [Index t](#) for time stacks.

 **Measuring on multi-channel images**

A multi-channel image is made up of individual fluorescence images. For multi-channel images you can choose to measure on each fluorescence image separately or to define one measurement object for all color channels simultaneously.


Clear the *Tools > Options > Measurement and ROI > General > Measure on all channels* check box.

Now, you will measure on each fluorescence image separately. To do so, set up the color channel you want on your monitor. To do this, use the navigation bar in the image window. Then carry out the measurement on this image. The measurement will be permanently linked to this color channel, i.e., the measurement will only be displayed on your monitor when the color channel on which you made this measurement is also on display.

The measurement results will be shown in the *Measurement and ROI* tool window. You can give every measurement the name of the color channel on which it was made. To do this, use the *Channel* measurement parameter.

Select the *Tools > Options > Measurement and ROI > General > Measure on all channels* check box.

Now, each measurement object you define will be measured on each color channel. All measurement results will be shown in the *Measurement and ROI* tool window.

 **Measuring on multi-layer images**

With some functions, e.g., with the *Image > Combine Color Images...* function, a multi-layer image will be created. This multi-layer image is made up of several layers. You can find more information on multi-layer images in the online help.

Measurements always apply to one image layer. For this purpose, show the image layer on your monitor, on which you want to make measurements. To do so, use the *Layers* tool window. Then carry out the measurement on this image layer. The measurement will be permanently linked to this image layer, i.e., the measurement will only be displayed on your monitor when the image layer on which you made this measurement is also on display.

The measurement results will be shown in the *Measurement and ROI* tool window. You can give every measurement the name of the image layer on which it was made. To do this, use the *Layer* measurement parameter.

00150 28052014

11.2.2. Measuring images


Your software offers a wide range of measurement functions. They enable you to measure distances and areas on an image quickly.

The following step-by-step instructions present the interactive measurement functions to you by way of several examples.

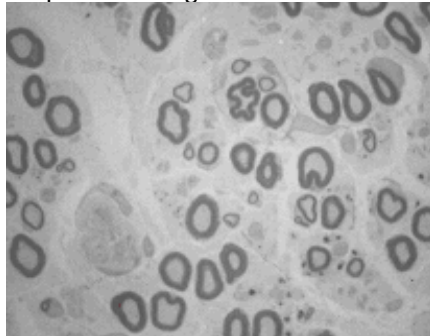
Measuring image objects interactively

Task

You want to measure the diameter of some cells. To do this, load a suitable image, or acquire one. Measure the diameter of some cells. Then edit the measurement and delete some of the measurements that have been performed. Enter the results in a MS-Excel sheet.

1. If necessary, use the *View > Tool Windows > Measurement and ROI* command to have the *Measurement and ROI* tool window displayed.
 - You'll find the tool window at the lower edge of the user interface. It could be under another tool window. Click the  *Measurement and ROI* tab at the bottom of the user interface to bring the tool window into the foreground.

Loading an image 2. Acquire an image or load one.



- During the installation of your software some sample images have been installed, too. You can follow these step-by-step instructions for measuring images when you use the exemplary image Neurons.tif.

Setting the labeling color

The measurement results will be written into the image according to the default settings, in red font color and without a background. This can be hard to read on some images. Change the labeling settings.

3. Use the *Tools > Options...* command.
4. Click the *Measurement and ROI > Measurement Display* entry in the tree view.
5. Click in the *Background Color* field, and choose, e.g., the color Black.
6. Select the *Text color > Fixed colors* option and select a suitable color from the palette. Select the color white to display the measurements in white and the labels in white on a black background.
7. Close the dialog box with *OK*.

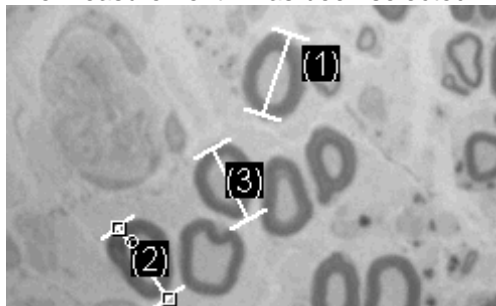
Measuring lengths



8. Click the *Arbitrary Line* button, located on the toolbar at the top of the tool window.
9. Click with your left mouse button at the starting point and end point of the reference distance.
10. If you have measured a reference distance, you can immediately proceed with the next measurement.
11. Click the *Arbitrary Line* button again to end the length measurement.
12. Take a look at the results in the tool window and in the image.



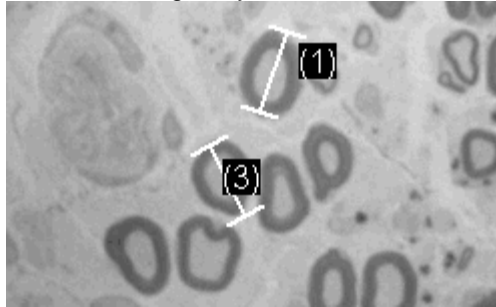
- The illustration shows the image with three executed measurements. The measurement 2 has been selected



Deleting measurements

13. Click one of the measurement results in the *Measurement and ROI* tool window.
 - The corresponding line will be selected in the image.
14. Press the [Del] key.
 - The measurement will be deleted both in the image and in the tool window.

- When a measurement has been deleted, the image and the tool window contain one measurement less. The IDs of the remaining measurements won't be changed by the deletion of a measurement.



Note: When you've completed the measurements, you should switch off the measurement mode, since otherwise, you might inadvertently select your measurements and move them.

Exporting results to
MS-Excel



15. Check whether one of the buttons on the *Measurement and ROI* tool window's toolbar appears clicked. Release this button
16. To do this, click the *Export to Excel* button.
17. In the In/Output dialog box you set up the directory in which the data is to be saved, and enter the name of the MS-Excel sheet. Adopt the file type *Excel-Sheet (*.xls)*.
18. Click the *Save* button to have the MS-Excel sheet with the measurement results saved.

Closing the image

19. Click the small button showing a cross [x], located at the right of the image name in the document group.
 - You have changed the image because you've added interactive measurements. For this reason, you'll receive a query whether you wish to save the image or not.
20. Save the image in the TIF or VSI file format. The measurements will then also be saved in the image file. They can at any time, be edited deleted or augmented.

Outputting various measurement parameters

Task

You want to measure some cells. Measure the cell as a circular object. Have a variety of measurement parameters, such as the area, the perimeter and the diameter, output. Have the diameter shown in the image.

Measuring areas

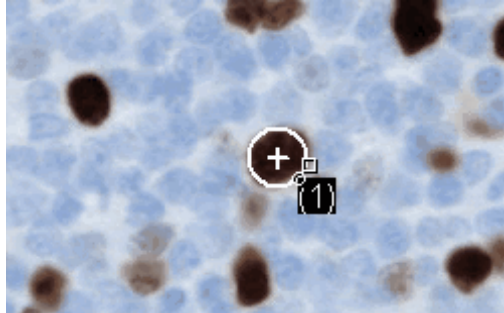


1. Acquire or load an image, the BadTissue.tif example image for example.
2. In the *Measurement and ROI* tool window, click the *2 Point Circle* button.
3. Leftclick the center point of a cell that you want to measure.
4. Move your mouse, and in the process drag out the circle. Match the circular object as well as possible to the cell. Click the left mouse button.
5. Click the *2 Point Circle* button again, and switch off the measurement mode.



6. Take a look at the result in the *Measurement and ROI* tool window.

- The illustration shows the image with a circle measured.



Viewing the list of measurement parameters



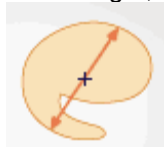
7. In the *Measurement and ROI* tool window, click the *Select Measurements* button.

- In the dialog box you'll see a list with all of the available measurement parameters. At the bottom of the dialog box you'll see a list of the measurement parameters that are currently calculated for all objects.
- A detailed description of this dialog box can be found in the online help.

Outputting additional measurement parameters

8. Go to the list of all of the available parameters, then click the *Diameter* measurement parameter.

- On the right, an illustration shows you how the parameter is calculated.



You can see that there are different ways in which the diameter of a 2D object can be calculated.

9. Click the *Mean* entry in the list under the illustration, to select the *Mean (Diameter)* measurement parameter. When you do this, the mean value of all of the possible diameters is determined.

10. Click the *Add 'Mean (Diameter)'* button.

- This measurement parameter will be added to the list of measurement parameters to be calculated. All of these measurement parameters will be displayed in the tool window.

11. Close the dialog box with *OK*.

12. Take a look at the result for the circle's diameter in the *Measurement and ROI* tool window.

Outputting measurement parameters in the image



13. Open the *Select Measurements* dialog box.

14. At the bottom of the list of all of the calculated measurement parameters, click the *Mean (Diameter)* measurement parameter.

15. To the right of this list you'll see a button with a blue arrow. Click this button to move the measurement parameter to the top of the list.

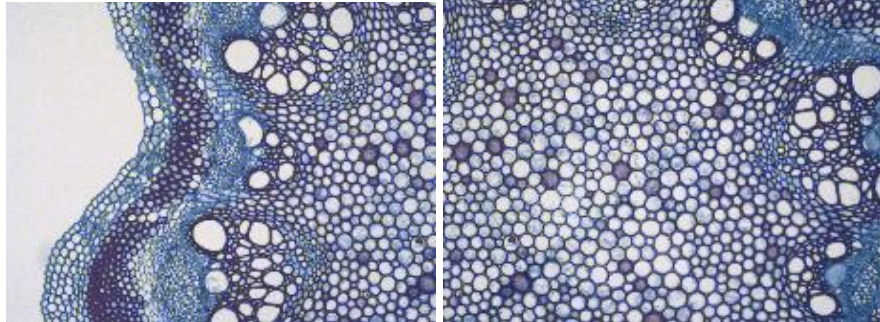
16. Close the dialog box with *OK*.

17. Take a look at the result for the circle's diameter in the image.

Measuring several images

Task You want to measure cells on multiple images. To do so, acquire some images and measure them one after another. Have the results from all images displayed simultaneously. Take a look at the mean value for all of the measurements.

Loading images 1. Acquire or load some images.



- During the installation of your software some sample images have been installed, too. You can carry out these step-by-step instructions using the Clematis04.tif and Clematis05.tif example images.

Measuring cells

2. Activate the first image in the document group.



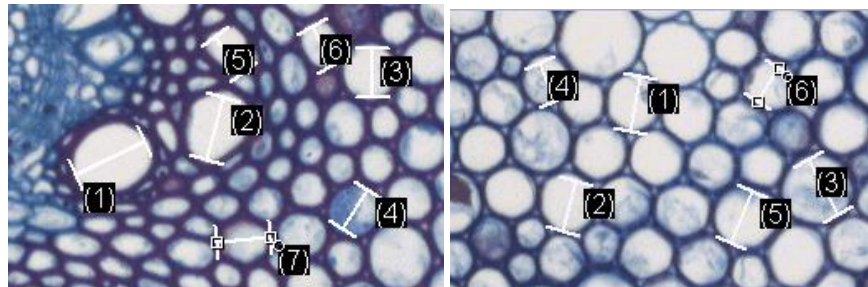
3. Click the *Arbitrary Line* button, located on the toolbar at the top of the *Measurement and ROI* tool window. Measure the diameter of several cells.

4. Activate the next image. Measure the diameter of several cells on this image, too.



5. Click the *Arbitrary Line* button again, and switch off the length measurement.

- Cells have been measured on both images.



Displaying the measurement results of all of the images



6. In the *Measurement and ROI* tool window, click the *Measurement and ROI Options* button.

7. Select the *Measurement and ROI > Results* entry in the tree view.

8. Clear the *Show measurement objects > Only of the active image* check box.

9. Close the dialog box with *OK*.

- Now the results for both images will be shown simultaneously in the tool window.
- Use the *Document* measurement parameter to display the name of the image with which the measurement results are associated in the results sheet. Now you can match the measurement results unambiguously to an image, even if all measurement results are displayed together in the tool window.

Viewing the statistical parameter

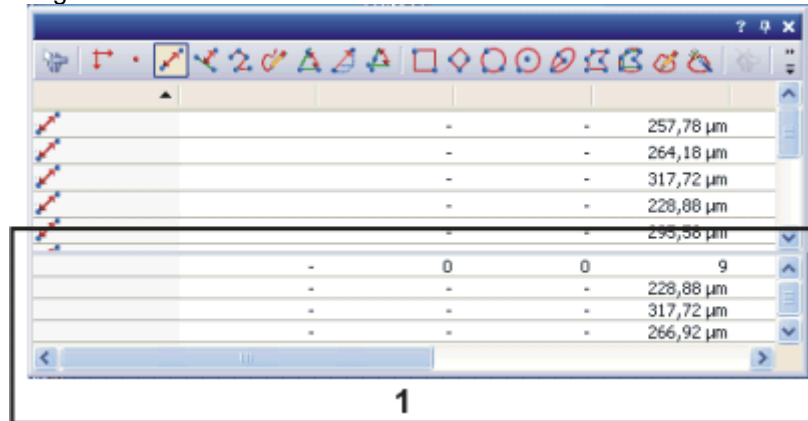


10. In the *Measurement and ROI* tool window, click the *Measurement and ROI Options* button.

11. Select the *Measurement and ROI > Results* entry in the tree view.

- In the *Statistic* group you'll find various statistical parameters.

12. Select the *Mean* check box.
13. Close the dialog box with *OK*.
 - Now, in the *Measurement and ROI* tool window under the measurement results, the chosen statistical parameter (1) will be shown. You can see there the mean value of the layer thickness for all of the measured images.



The screenshot shows a software window titled "Measurement and ROI" with a toolbar and a table of results. The table has several columns, with the last column displaying values in micrometers (µm). A red box highlights a row with a value of 9, which is labeled with a "1" below it.

				257,78 µm
				264,18 µm
				317,72 µm
				228,88 µm
				295,56 µm
		0	0	9
				228,88 µm
				317,72 µm
				266,92 µm

00154

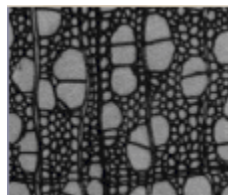
11.3. Carrying out an automatic image analysis

You can use an automatic image analysis to carry out numerous measurement tasks. Several typical tasks and their process flow are described here.

Note: Your software offers two different software packages for automatic object analysis. In the basic version, not all of the described functions are available. You can find more information in the individual step-by-step instructions.

11.3.1. Counting objects

Task You have an image with objects that interest you. You want to know how many of these objects there are in the image.



Preconditions The objects that you want to count must not be connected, but must be clearly separated from one another. The objects in the foreground should be optically clearly separated from the image's background. In the example image shown, the background is dark. The objects lie in the foreground and are light in color.

Preparations

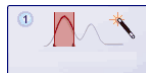
1. Use the *View > Tool Windows > Count and Measure* command to have the *Count and Measure* tool window displayed.
2. Acquire an image or load one.
 - During the installation of your software some sample images have been installed, too. You can immediately follow these step-by-step instructions when you use the exemplary image WoodVessels.tif.

Setting options



3. Open the *Options* dialog box by clicking the *Count and Measure Options* button, located in the *Count and Measure* tool window.
4. Select the *Count and Measure > Detection* entry in the tree view.
5. In the *Options* group, enter the value 5 in the *Minimum object size* field. An object must now be at least 5 pixels large in order to be counted as an object. By doing that, you will rule out the possibility that individual pixels, that may well have the same color or intensity as the objects, but don't belong to an object, are counted as objects, which would then falsify the results. This way you can exclude noise and dust particles.
6. Click *OK* to exit the *Options* dialog box.

Setting threshold values



7. In the *Count and Measure* tool window, click the *Automatic Threshold...* button to open the *Automatic Threshold* dialog box.
 - Should the *Automatic Threshold* button not yet be active, you will have to first activate it. To do so, select the *Automatic Threshold...* entry in the *Threshold* button's menu. You open this menu by clicking the small arrow next to the button.
 - The threshold values are set automatically in the *Automatic Threshold* dialog box.
 - All of the objects that have been detected will be displayed in color.
8. Check whether the objects have been correctly recognized. Should the objects not have been correctly recognized, go to the *Background*

group and enter whether the background is bright or dark.

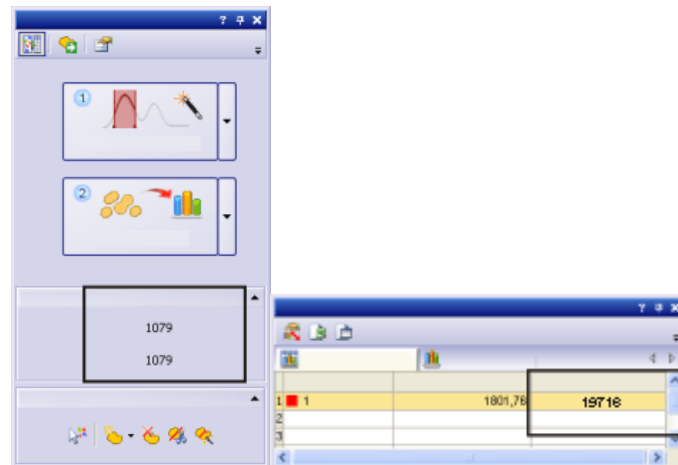
Select e.g., for the image shown above, the *Background > Dark* option, since the image shows bright objects against a dark background.

9. Only if the *Remove Phase* button in the *Phase* group is active: Delete all but one of the phases by continuing to click the *Remove Phase* button until the button becomes inactive.
 - By doing that, you will make certain that no phases from earlier analyses are still defined.

Outputting results

10. To obtain the results, click the *Count and Measure* button in the *Automatic Threshold* dialog box.
 - The *Automatic Threshold* dialog box will be closed.
 - The number of objects found is displayed in the *Object Count* group in the *Count and Measure* tool window.
 - The objects that have been analyzed are then displayed in color, on their own image layer. This image layer is called *Detected Objects*. Use the *Layers* tool window to make these image layers appear or disappear, or to delete them.

Prerequisite: You are working with the *Count & Measure Full* solution. In the basic version, the following functions are not available:




The number of objects detected will be shown below, in the *Count and Measure* tool window, in the *Object Count* group. Should you not be able to see this number, click the small black arrow to make it visible. If you have selected the *Object Count* measurement parameter, the number of objects will also be displayed in the *Count and Measure Results* tool window's results sheet.

Separating objects

Prerequisite: You are working with the *Count & Measure Full* solution. In the basic version you can't edit objects.

It is sometimes the case that two objects that are next to each other are not detected separately because, as far as the software is concerned, they are joined together. These sorts of objects can be separated manually.

1. Zoom into the image to enable you to better process the object.
2. Then click the *Manually Split Objects*  button, located in the *Edit Objects* group, then move your mouse pointer onto the image.
3. Now define a separation line through the object by clicking the left mouse button. Make sure, when you do this, that you drag the line over the object's outside edge, since otherwise it won't be separated.

4. Right click to confirm the separation line.
 - The object will then be divided up into two independent objects. The results will be updated.



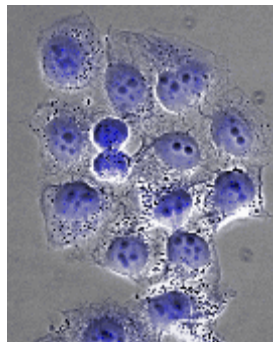
Left: Two objects are touching each other and thus are counted as a single object.

Middle: The joined up object has been selected.

Right: The joined up object has been separated, there are now two independent objects.

11.3.2. Counting object belonging to different phases (setting the thresholds)

Task You have an image on which you define two phases. You want to know how many objects there are per phase, in the image.



In the image, two phases are to be defined. The first phase is to map the black objects within the blue, round object. The second phase is to map the blue, round objects.

Preconditions The objects that you want to count must not be connected, but must be clearly separated from one another. The objects in both of the phases must have different intensity values by which one can differentiate between them.

Setting options



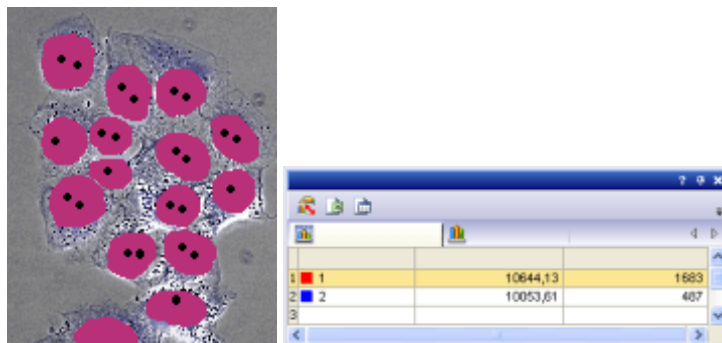
1. In the *Count and Measure* tool window, click this button, to open the *Options* dialog box.
2. Select the *Count and Measure > Detection* entry in the tree view.
3. In the *Options* group, enter the value 5 in the *Minimum object size* field to specify the minimum object size. By doing that, you rule out the possibility that individual pixels, that may well belong to the phase, but not to an object, are counted as objects, which would then falsify the results.
4. Select the *Count and Measure > Measurements* entry in the tree view.
 - In the basic version:
From the *Class Measurements* list, select the *Object Count* and *Object Class* entries.
 - With the *Count & Measure Full* solution:
Click the *Select Class Measurements...* button located in the *Measurements* group. In the *Select Class Measurements...* dialog box, add the *Object Count* and *Object Class* measurement parameters and close the dialog box.
5. Click *OK* to exit the *Options* dialog box.

Setting threshold values

6. In the *Count and Measure* tool window, click the *Manual Threshold...* button to open the *Manual Threshold* dialog box.
 - Should the *Manual Threshold* button not yet be active, you will have to first activate it. To do that, select the *Manual Threshold...* entry, in the *Threshold* button's context menu. You open this menu by clicking the small arrow next to the button.
7. Only if the *Remove Phase* button in the *Phase* group is active: Delete all but one of the phases by continuing to click the *Remove Phase* button until the button becomes inactive.
 - By doing that, you will make certain that no phases from earlier analyses are still defined.
8. Double click the *Phase Name* field and assign a name for the first phase. Click any position outside this field, or click the [Enter] key to leave the field again.
 - The first phase in the *Phase thresholds for channel '...'* group will be automatically selected.
9. Click the *New Threshold* button to set an initial value for the selected phase's threshold value range.
 - As soon as you move your mouse pointer onto the image it will change its shape to that of a pipette.
10. Click on one pixel or on the image area whose intensity value is to be utilized as the initial value for the threshold range.
 - Once the initial value has been set, your mouse pointer will automatically change into a pipette with plus icon.
11. Then, continue clicking pixels that are typical of the first phase, until the required structures in the image are a part of the phase.
12. Should too many pixels have been selected, click the *Shrink Threshold* button to have these pixels excluded from the phase again.
 - The threshold value range will continue to be reduced until it no longer contains the pixels you have selected.
 - Alternatively, click the *Undo Pipet* button.
13. Click the *Add Phase* button to add the second phase, then proceed exactly as you did for the first phase.

Outputting results

14. To obtain the results, click the *Count and Measure* button in the *Manual Threshold* dialog box.
 - The *Manual Threshold* dialog box will be closed.
15. Open the *Count and Measure Results* tool window by using the *View > Tool Windows > Count and Measure Results* command.



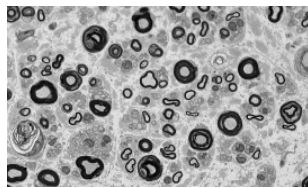
The total number of objects detected in all of the phases will be shown below, in the *Count and Measure* tool window, in the *Object Count* group. The results for

the *Object Class* and *Object Count* measurement parameters, that's to say, the sum of the objects per phase, will be displayed in the results sheet. Furthermore, you will recognize the phases by the colors that have been assigned to them. You can compare the results for both of the phases directly with each other.

11.3.3. Measuring objects (selecting and outputting measurement parameters)

Prerequisite: You are working with the *Count & Measure Full* solution. In the basic version you can't measure individual objects.

Task You have an image with objects of different sizes. You want to know the area of the largest object and to have a close look at that object in the image. In addition to that, you want to export the results into a sheet.



Preparations

1. Acquire or load an image.
2. Carry out an automatic object analysis on the image.
3. Open the *Options* dialog box by clicking the *Count and Measure Options* button, located in the *Count and Measure* tool window.
4. In the tree view, select the *Count and Measure > Measurements* entry, then click the *Select Object Measurements...* button, located in the *Measurements* group.
5. In the *Select Object Measurements* dialog box, add the *Area* and *Object ID* measurement parameters and close any open dialog boxes.
6. Next, in the *Count and Measure* tool window, click the *Count and Measure* button to output the results.

Selecting a measurement parameter



Viewing and sorting the results

7. In the *Count and Measure Results* tool window, select the *Object Measurements* results view.
 - The measurement values for the objects' areas are displayed in the *Area* column.
8. Sort the *Area* column to find out which value is the smallest or the largest. To do so, double click on the header of the *Area* column.
 - This column's measurement values will then be sorted in ascending or descending order.
9. Double click the header of the column again to sort the measurement values in the reverse order.
 - An arrow in the header will show you the direction in which they are sorted.

Object - sheet link

10. Select the largest value in the *Area* column.
 - The corresponding object will likewise be selected in the image window. In this way, you can easily find an object that belongs to a specific value, and view it.

Exporting the results to a sheet



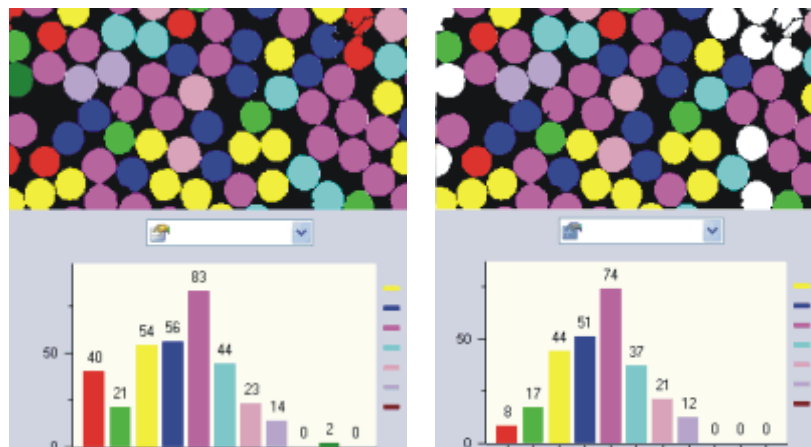
11. In the *Object Measurements* results view, click the *Export to Excel* button.
12. In the *Export Object Results* dialog box, assign the sheet a significant name, then save it in the required directory.

11.3.4. Filtering objects

Prerequisite: You are working with the *Count & Measure Full* solution. In the basic version you can't filter objects.

Objects that disturb you, or that don't interest you, can be excluded from the measurement results. All of the measurement values that lie outside the defined measurement value area, won't be displayed, nor taken into account in any of the results views.

Task On an image with spheres of different sizes, 9 size classes are defined. You want to know how many spheres fall into which size class. When the analysis has been carried out, you discover that the number of the small spheres has been overestimated, because spheres that weren't correctly separated, were also taken into account (image on the left). Define an object filter that only counts roughly circular objects.



Left: At the top right of the image, you can see some spheres that weren't divided properly. They have been sorted into the class of small spheres and are displayed in red.

Right: After the definition of an object filter, the number of objects in each class has changed. In particular, the red class of small spheres now has fewer objects.

- Preparations**
1. Load the image you want to analyze or acquire one.
 2. Carry out an automatic object analysis on the image.
 3. In the *Count and Measure Results* tool window, switch to the *Object Filter* results view.
 - In the table you will see a list of all of the selected measurement parameters and their corresponding filter ranges. There will always only be one measurement parameter active.
- Entering the filter range directly**
4. In the table, click the measurement parameter for which you want to define a filter range.
 5. Double click in the *[Min.]* field, located next to the measurement parameter to enter the lower value for the filter range.
 6. Either enter the required measurement value directly, or use the arrow keys.
 7. Double click in the *[Max.]* field, then enter the higher value for the filter range.
 - The higher value itself no longer belongs to the filter range.
 - You can delete individual values by double clicking the value, then pressing the [Del] key.
- Defining the filter range interactively**
8. In the table, click the measurement parameter for which you want to define a filter range.



9. Click the *Select minimum range* button, above the *Measurement* list, to define the filter range's lower value.

- The mouse pointer will change its form.

10. Click an object whose measurement value is to be used as the lower value for the filter range.

- The measurement value will then be automatically adopted in the *[Min.* field. When you, for example, define a filter range for the *Area* parameter, click the smallest object that you still want to measure.
- In the image window, the result of the filtering of the objects can be seen straight away. All of the values that are outside the defined filter range will be excluded from the results.
- The filter range contains precisely those values that are to appear in the measurement results. All of the values that are outside the defined filter range will be excluded from the results.



- The *Toggle Object Filter* button appears clicked, thereby showing you that the object filter is active.



11. If you want to undo the selection you've made, click the *Clear minimum range* button.



12. Click the *Select maximum range* button to define the filter range's upper value.

13. Click an object whose measurement value is to be used as the upper value for the filter range. Click the largest object that you still wish to measure.

- The measurement value is rounded up and automatically adopted in the *Max.]* field. The object is still within the filter range.

Switching off the Object Filter



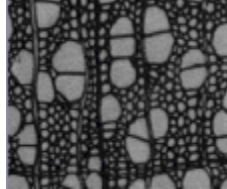
14. Release the *Toggle Object Filter* button.

Note: A defined object filter is not automatically deactivated when you load another image. If, for example, no objects are shown, make sure that the object filter is deactivated.

11.3.5. Classifying objects

Prerequisite: You are working with the *Count & Measure Full* solution. In the basic version, you can't define your own classifications.

Task You have an image with two object classes, e.g., large and small cells. You want to know how many objects fall into which size class.



Preparations

1. Acquire an image or load one. You can follow these step-by-step instructions using the *WoodVessels.tif* example image .

2. Perform an automatic object analysis on the image.

3. Select the *Area* object measurement.

Selecting measurement parameters for the object classes

4. In the *Count and Measure Results* tool window, select the *Class Measurements* results view.

5. Click the *Select Class Measurements* button, then in the *Select Class Measurements* dialog box, add the *Mean (Area)*, *Object Class* and *Object Count* measurement parameters.

- With the *Mean (Area)* parameter, the mean area of all of the objects in a class will be calculated. That's to say, the parameter give you a measured value for how large the objects in this class are, on average.
- With the *Object Class* parameter, you write the name and the color of the class in the results sheet, as well. You should, without fail, adopt this parameter in the results sheet to make it possible to assign the measurement results correctly to the individual classes. You can also adopt this parameter in the *Object Measurements* results sheet. Then, in the results sheet, you'll be able to immediately recognize to which class each of the individual objects belongs.
- Finally, the *Object Count* parameter supplies you with the values that were sought in the tasks set: the number of objects that are in each class.

6. Close the *Select Class Measurements* dialog box.

Defining classes

7. Open the *Options* dialog box by clicking the *Count and Measure Options* button, located in the *Count and Measure* tool window.

8. Select the *Count and Measure > Classification* entry in the tree view.

9. In the *Current Classification* group, click the *New Classification* button, then select the *New 'One parameter Classification'* entry.

- The *Define 'One parameter' Classification* dialog box opens.

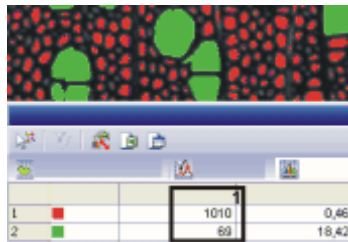
10. Enter a descriptive name for the new classification in the *Name* field, *size class* for example.

11. Select the *Area* entry in the *Measurement* list.

12. Click the *Automatic Classification* button to switch to the *Automatic Classification* dialog box.

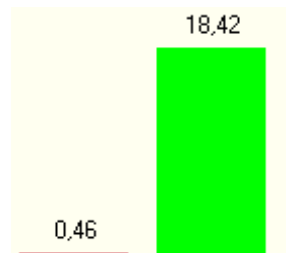
13. In the *Automatic Classification* dialog box, click the *Get Min./Max. from Image* button. Then the smallest and largest value of the selected parameter, that has been entered in the *Minimum* and *Maximum* fields, will be used.

- In this way, you'll be certain that all of the objects in the image can be assigned to one of the classes that have been defined.
14. Enter the value 2 in the *Number of classes* field, and in the *Scale* field, select the *Logarithmic* entry.
- By doing this, you have defined two size classes.
- Outputting results 15. Click *OK* and then the *Count and Measure* button, located in the *Define 'One parameter' Classification* dialog box.
- The classes will be displayed in the image in color. The measurement parameters that have been selected for the classes will be output in the *Class Measurements* results view.



In the illustration, you can see the image with both of the size classes. The column (1) shows the number of large (green) and small (red) cells that was being looked for.

16. Close the *Define 'One parameter' Classification* dialog box.
- In the *Options > Count and Measure > Classification* dialog box, the new classification is active in the list. You can now use this classification for other analyses as well.
17. Close the *Options* dialog box with *OK*.
18. Then, in the *Count and Measure Results* tool window, activate the *Class Histogram* results view to have the class results displayed as a bar chart.



In the illustration, you see the results for the object classes in the *Class Histogram* results view. The mean area ratio for the object classes is displayed as a diagram. You can clearly see that the green objects are significantly larger than the red objects.

00513

12. Running experiments

12.1. Overview

What exactly is the Experiment Manager?

You can use your software to implement complex acquisition processes. Use the Experiment Manager to define and run complex experiments involving image acquisition with your software. You can always re-use existing experiment plans or adapt them to new conditions. You can acquire multi-dimensional images during the experiment. If your microscope has motorized hardware components, you can control these with the software during the experiment. Or you can use an RTC (Real Time Controller), to use external devices in your experiments.

The Experiment Manager is a graphic Process Manager

The idea behind the Experiment Manager

You can create a graphic experiment plan with the *Experiment Manager* tool window. This experiment plan contains a series of commands, the acquisition of images for example, that are carried out one after the other.

Example: Use the Experiment Manager to acquire several multi-channel fluorescence images of a certain position on the sample at certain intervals.

How the Experiment Manager differs from the Process Manager

Just like the Experiment Manager, you can use the *Process Manager* tool window to handle complex acquisition processes. The Experiment Manager can be used as an alternative to the Process Manager. It's more intuitive for more complex processes and offers you more options.

- In the *Experiment Manager* tool window, you can trigger a particular device at a particular time using the RTC to add a chemical to your sample, or to heat or illuminate your sample, for example.
- Only the Experiment Manager allows you to define loops within loops. This makes it possible to repeat a fast time stack several times at particular intervals.
- The Experiment Manager allows you to use streaming to attain shorter intervals between two separate image acquisitions in a time stack than are possible with the Process Manager.

Prerequisites for using the Experiment Manager

Prerequisite: The *Experiment Manager* tool window is only available with the highest software package.

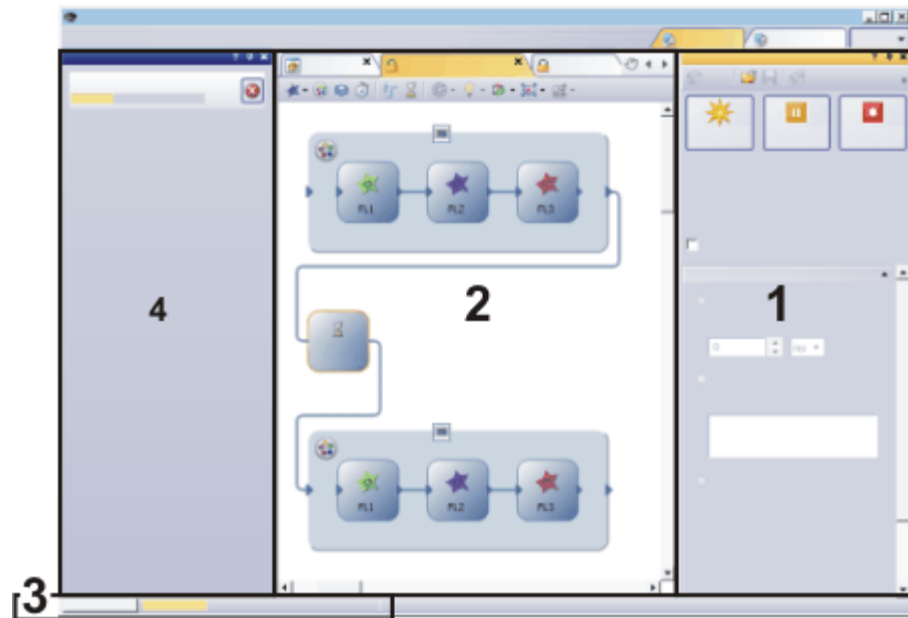
The system has been configured. The observation methods have been defined.

Make sure that your software is correctly configured.

When you want to acquire multi-channel fluorescence images, it makes sense to define observation methods for your color channels before you define the experiment. Only when you've defined an observation method can you assign a fluorescence color to the individual color channels when acquiring fluorescence images, for example.

Experiment Manager's user interface

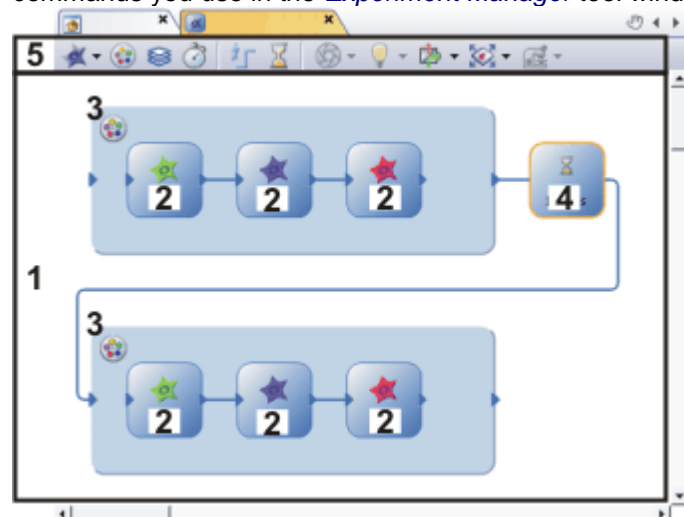
The Experiment Manager is composed of the *Experiment Manager* tool window and an experiment plan.



The Experiment Manager is composed of the *Experiment Manager* tool window (1) and an experiment plan (2). When you run an experiment, a progress bar is displayed on the left of the status bar at the bottom of the monitor (3). When an experiment is running over a long period of time, you can also display the *Task* tool window (4) to get more information about the progress of the experiment.

Use the *Experiment Manager* tool window to create an experiment plan, to edit an existing experiment plan or to start an experiment plan. To keep a good overview of the flow chart in the experiment plan, specify the settings for the commands you use in the *Experiment Manager* tool window.

The experiment plan



The illustration shows the elements on the user interface that belong to the experiment plan. The experiment plan is a document that is displayed in exactly the same way as images and other documents in your software's document group.

A document of the *Experiment plan* type is essentially a canvas on which you define the experiment by creating a flow chart (1).

An experiment plan contains particular commands, concerning image acquisition, as a rule. Each command is graphically displayed in the experiment plan with its own icon. The example displayed above contains the icons for a fluorescence image acquisition (2), the icons for the creation of a multi-channel image (3) and the icon for the *Wait* command (4).

The commands are connected with a line while an arrow clearly defines the order of the commands.

Experiment plans have their own toolbar in the document window itself (5). You can find all the commands that you can use in the experiment on this toolbar.

You can find the description of the toolbar in the online help.



Activating experiment plans in the document group

Note: When you're running an experiment, the acquired images are displayed in the document group, as a rule. In doing so, they hide the experiment plan.

If you are working in the experiment plan, click the *Keep Experiment Visible* button. You can find this button on the *Experiment Manager* tool window's toolbar. Now the experiment plan will be shown in its own document group. The document group will automatically appear when you start the experiment. This way you make sure that the experiment plan remains visible so that you can work on it easily.



Reducing the size of the experiment plan

If your experiment plan contains a lot of commands, you can reduce its size to maintain an overview.

Use the *Zoom Out* and *Zoom In* buttons on the *Zoom* toolbar. The zoom factor of the experiment plan is displayed on the *Zoom* toolbar. The maximum zoom factor is 100%.

Alternatively, activate the experiment plan. Hold the [Ctrl] key. Now you can use the mouse wheel to zoom in and out of the experiment plan.

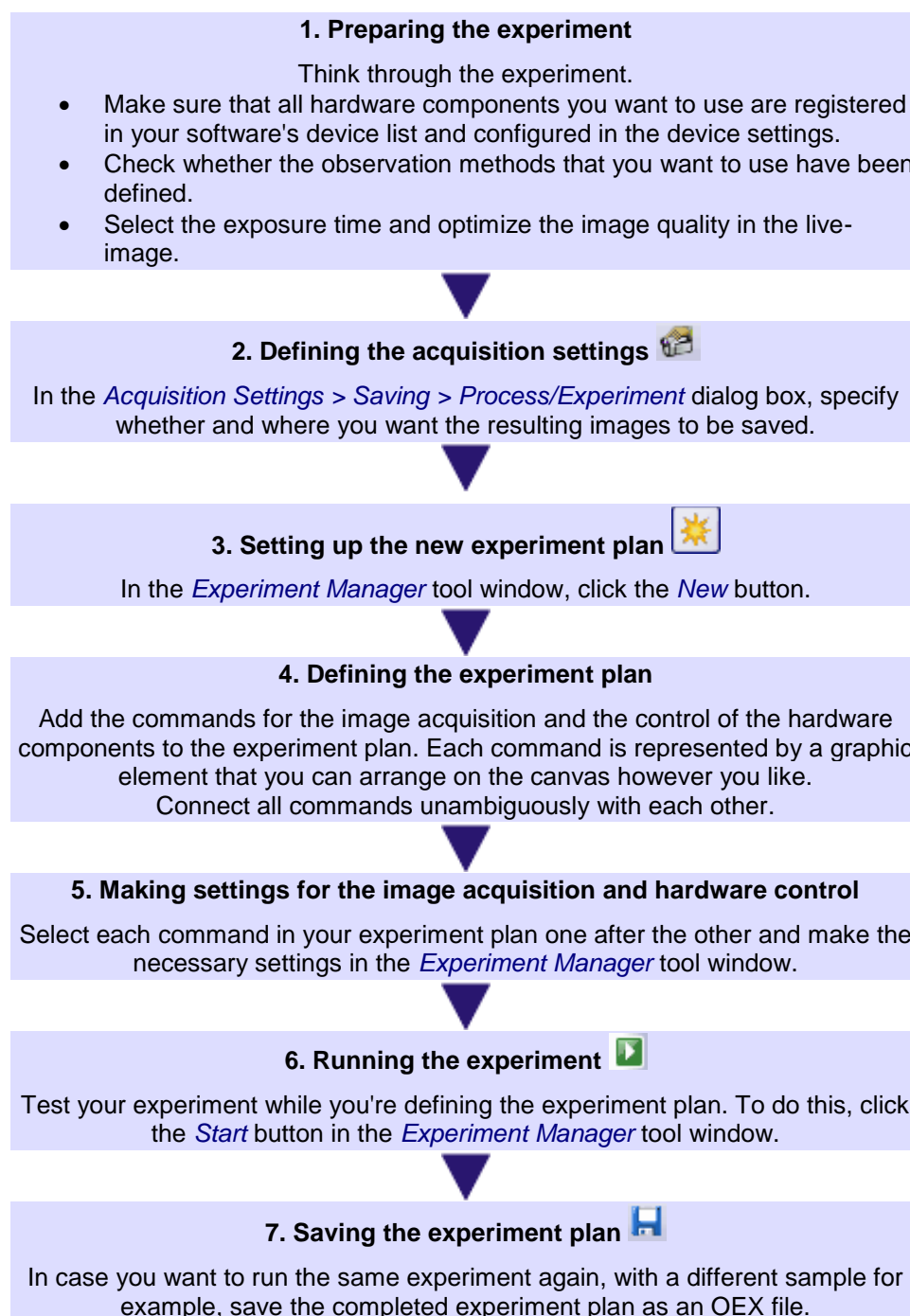
General process flow

You can use the Experiment Manager for two different types of tasks.

- Defining and running a new experiment
- Using an existing experiment plan

Defining and running a new experiment

The following process flow chart displays the basic steps of the process.



Using an existing experiment plan

You can load an existing experiment plan at any time and run the experiment again. The following process flow chart displays the basic steps of the process.

1. Defining the acquisition settings

In the *Acquisition Settings > Saving > Process/Experiment* dialog box, specify whether and where you want the resulting images to be saved.

2. Loading the experiment plan

Load a OEX file in the document group.

3. Adjusting the experiment plan

Select each command in your experiment plan one after the other and make the necessary settings in the *Experiment Manager* tool window. Change the exposure time for the acquisition of fluorescence images, for example.

4. Running the experiment

In the Experiment Manager tool window, click the *Start* button.

5. Saving the experiment plan

Decide whether you want to save the changed experiment plan.

00800 14072014









12.2. Toolbar - Experiment plan







The experiment plan is a document that is displayed in exactly the same way as images and other documents in your software's document group. The experiment plan contains its own toolbar with all the commands that you can use in an experiment plan.

Adding commands to an experiment plan

Click a button on the experiment plan's toolbar to select the corresponding command. Now you can insert the selected command in the experiment plan by clicking on the canvas.

The following table lists the buttons which are available on the toolbar.

	Image Acquisition	The image acquisition is the basis for every experiment. Click this button to add an image acquisition to the experiment.
	Multichannel Group	You can combine a series of fluorescence images into a multi-channel image.
	Z-Stack-Loop	Add a Z-stack acquisition to your experiment.
	Time Lapse Loop	Add a time stack acquisition to your experiment.
	Stage Loop	Move the stage to different positions on the sample during the experiment. You can use the position list that you defined in the <i>Stage Navigator</i> tool window, or you can define your own positions for the experiment on the sample's overview image.
	Digital Port	Trigger a device remotely before or after the image acquisition. You can use this to add a chemical or to illuminate your sample with light of a particular wavelength or intensity, for example.
	Wait	Integrate a delay between two image acquisitions.
	Ratio Intensity profile	<p>You can use the following evaluation methods in your experiment, to analyze the acquired images straight away. Click the small arrow next to the button to open a menu. The button for the last used command is displayed on the toolbar.</p> <p>You can use the <i>Ratio</i> command to measure the change in the concentration of calcium ions in a time stack, for example.</p> <p>Use the <i>Intensity Profile</i> command. An intensity profile shows how the intensity within one, or within several image segments (ROIs), changes over a period of time or over the different Z-positions.</p>

	Hardware components	<p>Usually various different devices, such as a camera and microscope, belong to your system. These hardware components can be controlled with your software in some systems. You can use these hardware components in an experiment and open or close a shutter, for example.</p> <p>Prerequisite: A hardware component can only be controlled by your software during an experiment when it's been correctly registered and configured in your software.</p>
	IX3 FRAP	When you use a FRAP system, you can bleach particular areas on your fluorescence samples with a laser.
	TIRF	When you use a TIRF system, you can define the laser positions for the TIRF illumination and thus the penetration depth in one command.
  	Move XY Move Z Autofocus	<p>Your software can move the stage between the acquisition of two images. Click the small arrow next to the button to open a short menu with the following commands for controlling the stage. The button for the last used command is displayed on the toolbar.</p> <p>Use the <i>Move XY</i> command to specify a different position on the sample.</p> <p>Use the <i>Move Z</i> command to move the stage up or down.</p> <p>Use the <i>Autofocus</i> command to focus the sample before the image acquisition.</p>

00803 09072014

12.3. Working with the Experiment Manager

Use the Experiment Manager to define and run complex experiments involving image acquisition and image analysis with your software.

The following instructions guide you step by step through the definition of a typical experiment. The complexity of the experiments described increases from example to example.

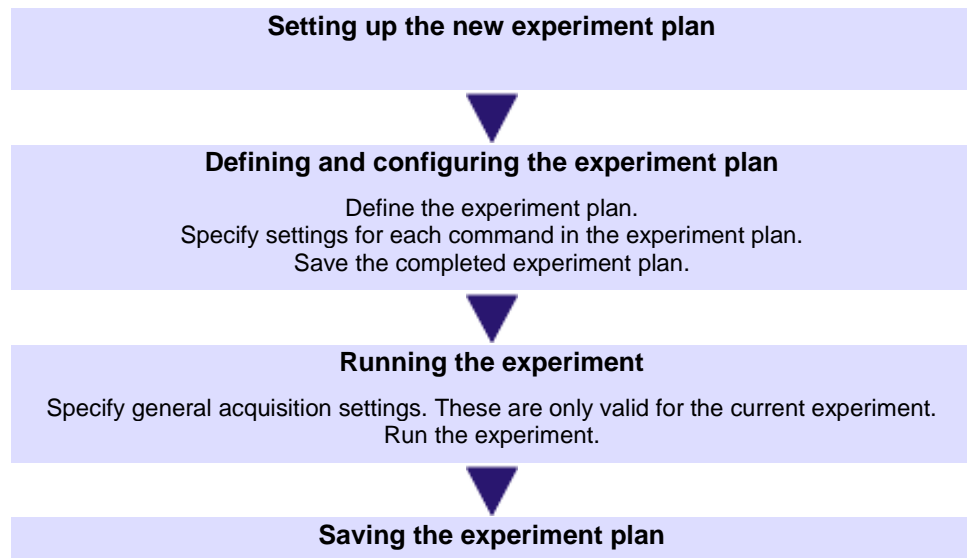
12.3.1. Defining a new experiment

Acquiring fluorescence images

Task Your sample has been stained with the DAPI, FITC and TRITC fluorochromes. Define an experiment plan for acquiring several fluorescence images and run the experiment.


- Preconditions*
- The system has been configured.
 - You have defined suitable observation methods for your color channels.

The following process flow chart displays the basic steps of the process.

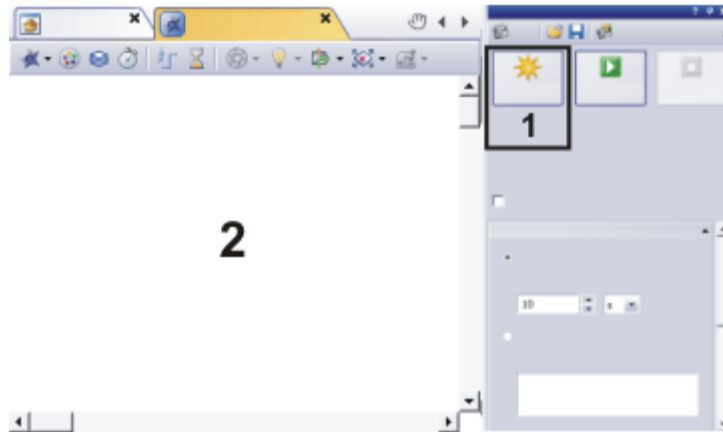


Setting up the new experiment plan



1. If necessary, use the *View > Tool Windows > Experiment Manager* command to show the *Experiment Manager* tool window.
2. In the *Experiment Manager* tool window, click the *New* button to create a new experiment.
 - This automatically creates a new document of the *experiment plan* type in the document group.
 - The experiment plan contains its own toolbar with all the commands that you can use in an experiment plan. Exactly which commands appear on the toolbar depends on your system configuration.
 - In the document group, experiment plans are identified by this icon  in the header.
 - The default name for the experiment plan is *Experiment_<sequential No.>*. You can change the experiment plan's name to anything you want when saving it. A small asterisk next to the name indicates that the document hasn't been saved yet.

Please note that the name of the experiment plan isn't linked to the experiment name that you enter in the *Experiment Manager* tool window. The entry in the *Experiment name* field is, by default, incorporated into the names of the images that you acquire with the experiment plan.



Click the *New* button (1) to create a new empty experiment plan (2).

Defining the experiment plan



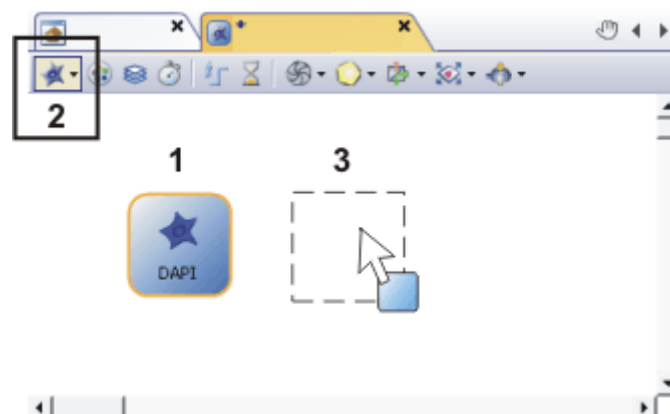
3. Define the first image acquisition command. Click the small arrow next to the *Image Acquisition* button to open a menu. Select the observation method that you want to use for the first image acquisition, *DAPI* for example.

- All of the observation methods that have been defined in your software are listed in the menu.

Note: When an experiment starts, the currently set objective and the camera are used during the whole experiment. If you have defined settings for the camera or the objective in an observation method and you then add this observation method to an experiment plan, these settings are not automatically adopted in the experiment.

Select the camera and the objective you want before starting the experiment. To do this, select the corresponding observation method in the *Observation Methods* group in the *Microscope Control* tool window.

4. Click in the canvas on the position where you want to place the image acquisition command with the DAPI observation method in the experiment plan.



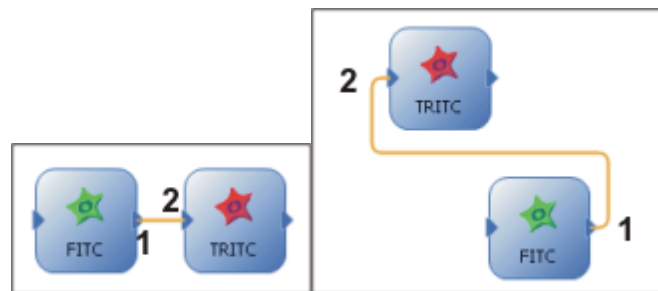
The experiment plan already contains the image acquisition command with the DAPI observation method (1).

Click the *Image Acquisition* button to add another image acquisition command to the experiment plan (2). The button corresponding to the selected command becomes active. You can recognize this status by the button's colored

background.

You can now move the image acquisition command on the experiment plan with your mouse (3).

5. Add the other two commands for image acquisition with the FITC and TRITC observation methods as well and arrange the commands in a row.
 - The three commands have to be connected to each other with a line. This line has an arrow which unambiguously defines the order of the commands. Depending on where you placed the commands in relation to each other, they may already be connected.
 - The experiment plan is constantly checked for errors by default. If the two image acquisition commands aren't connected, a yellow warning sign ⚠ appears at the top left of the experiment plan.
 - You can also switch off the automatic checking of the syntax. To do so, use the *Acquisition Settings > Acquisition > Experiment* dialog box.
6. You can define the connector between two commands in the experiment plan manually. To do so, create a connector by dragging one of the control points on the edge of a command to a control point on the following command. When doing this, you always have to connect an output control point (on the right of the command) with an input control point (on the left of the command).
 - Your software automatically draws the connector in the experiment plan. When you move commands in the experiment plan, the connector between these commands is automatically adjusted.



Both experiment plans show the same experiment. First, an image is acquired with the FITC observation method and then an image with the TRITC observation method. The green *FITC* command's output control point (1) is correctly connected to the red *TRITC* command's control point (2).

The connector has been selected in both experiment plans and is therefore displayed in color.

Configuring the experiment plan

Define the exposure time and other acquisition settings for the image acquisition commands.

7. Select the DAPI command in the experiment plan.
 - In the *Experiment Manager* tool window, the *Image Acquisition*, *Camera Settings*, and *Display* groups are now shown.
8. Click the *Send Settings* button. You can find the button in the *Image Acquisition* group in the *Experiment Manager* tool window.
 - The DAPI observation method is specified on the microscope.
9. Click the *Live* button in the toolbar at the top of the *Experiment Manager* tool window to switch to live mode. In the live-image, check the exposure time and focus on the sample.
 - Note: The live-image covers the experiment plan in the document group. When you turn off live mode again, the live-image is closed by default



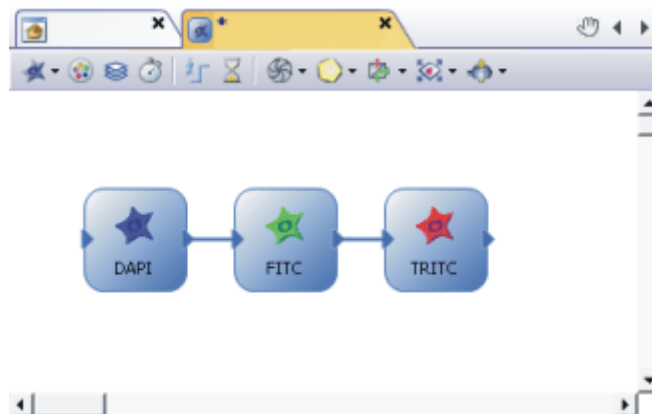
and you see the experiment plan again.

If you've selected a different setting for the live-image, you can activate the experiment plan after ending the live mode, in the *Gallery* tool window, for example.

- Set the resolution and the bit depth in the *Experiment Manager* tool window. Set the exposure time and the sensitivity or gain for an optimal image.



- Note: You can also use the *Camera Control* tool window to optimize the live-image. The acquisition settings in the Camera Control tool window are not, however, transferred automatically to the selected command in the experiment plan. To do this, click the *Get Settings* button in the *Experiment Manager* tool window .
- In the same way, define the acquisition settings for the FITC and TRITC commands. You can make different settings for each image acquisition command. For example, expose the different fluorescence images differently to even out the differences in the light intensity.



The experiment plan contains three individual image acquisition commands. Because the image acquisition commands are linked to an observation method, before the image acquisition your system automatically takes on the settings you defined in the observation method . For the acquisition of fluorescence images, the required mirror turrets are brought into the light path, for example. The experiment will produce three fluorescence images.

Running the experiment

Specify some general acquisition settings for the running of the experiment. These acquisition settings are not saved together with the experiment plan.



- Click the *Acquisition Settings* button, located in the *Experiment Manager* tool window's toolbar.
- Select the *Saving > Process/Experiment* entry in the tree view. Here, you specify whether and how the acquired images are to be automatically saved. You can have the acquired images saved in a database or in a directory of your choice. You can also switch off the automatic save process. In this case, the acquired images stay open in your software's document group after the experiment has finished.
- Select the *Document Name > Process/Experiment* entry in the tree view. Here, you specify how the acquired images should be named.
- You set some camera settings, like Online Deblur for example, globally for the whole experiment in the *Camera Control* tool window. Just like the acquisition settings, they're not saved together with the experiment.
- Click the *Keep Experiment Visible* button, located in the toolbar at the top of the *Experiment Manager* tool window.

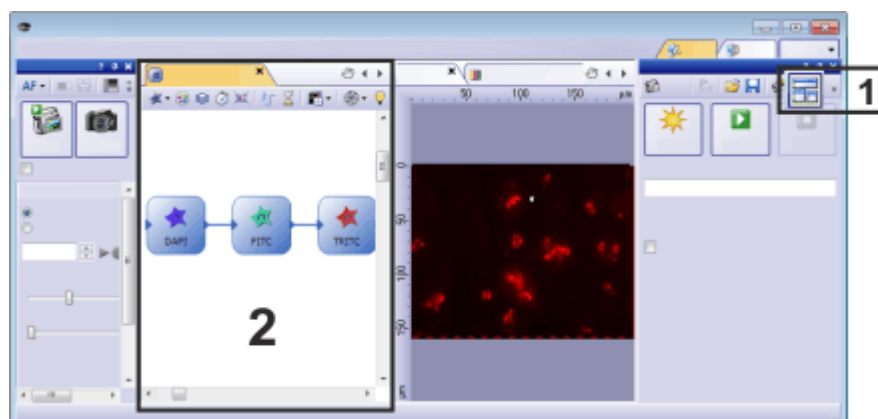


- The button is active when this mode is active. You can recognize this status by the button's colored background.



17. Click the *Start* button, located in the *Experiment Manager* tool window, to run the experiment.

- Note: You can only start the experiment when the experiment plan is active in the document group. If another document is active, an image for example, use the *Gallery* tool window to activate the experiment plan in the document group.
- The experiment starts immediately. Three individual fluorescence image will be acquired.
- When the *Keep Experiment Visible* button is active, a new document group will automatically be created in the experiment plan after the experiment is started. Now the experiment plan remains visible while the experiment is running.
- When the images acquired by your experiment are automatically saved, the experiment plan will be automatically saved, too. The automatically saved experiment plan is named like the first image that has been acquired.



When the button (1) is active, a new document group (2) is automatically created for the experiment plan.

Saving the experiment plan

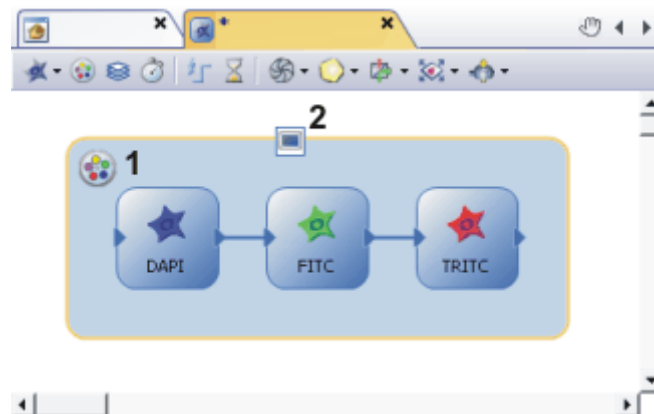
18. Activate the experiment plan in the document group.
19. Use the *File > Save As...* command to save the experiment plan. Save the experiment plan, under the name *3_FL_Images*, for example.
 - An experiment plan will be saved in the OEX file format.
 - If you have the results of the experiment automatically saved, the experiment plan is also automatically saved so that the experiment will be as well documented as possible. You can find the saved experiment plan in the same directory as the other data is saved in. You can specify the storage location in the *Acquisition Settings* dialog box.

Acquiring multi-channel fluorescence images

Task Your sample has been stained with the DAPI, FITC and TRITC fluorochromes. Define an experiment plan for acquiring a multi-channel fluorescence image. Run the experiment and acquire a multi-channel image.

1. Load an experiment plan in which three fluorescence images are acquired one after the other.
2. Use the *File > Save As...* command to save the experiment plan with a new file name.
3. Click the *Multichannel Group* button. You can find the button on the toolbar at the top of the experiment plan.
4. Draw a frame around the fluorescence image acquisition commands. All the commands in the multi-channel group will automatically be combined into a multi-channel image after the acquisition.

Adding a multi-channel group



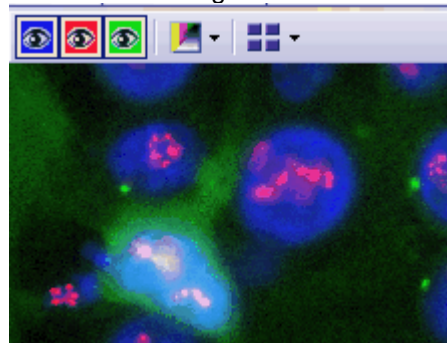
The illustration shows an experiment plan for a multi-channel image acquisition. The name of the multi-channel group (1) and the status of the online display (2) are displayed in the experiment plan.

The experiment will produce a multi-channel image with three color channels.



5. Test the experiment. To do this, click the *Start* button in the Experiment Manager tool window.
 - The acquisition of the multi-channel fluorescence image starts immediately.
 - The acquisition has been completed when you can again see the *Start* button in the *Experiment Manager* tool window.

The acquired fluorescence image is displayed in the image window by default after the experiment is finished. In the image window, all three color channels are superimposed on each other so that you see all three fluorescence images at the same time.



- In the image window, take a look at the multi-channel fluorescence image that has been acquired. If necessary, change the settings for individual commands in the experiment plan.

Defining a Z-Offset for individual color channels

Usually, the focus position is different for each color channel. Extend your experiment plan and select the optimal focus position for each color channel.



You acquire a multi-channel fluorescence image with three color channels with this experiment plan.

- Select a color channel in the experiment plan.
- Choose the observation method that belongs to the selected color channel and switch to live mode.
- Bring the sample into focus.
- Select the *Use Z-Offset* check box in the *Experiment Manager* tool window. The check box is located at the top of the tool window.
 - The *Z-Offset* group at the bottom of the *Experiment Manager* tool window is now available for image acquisition commands.
- Click the *Define Z Reference* button in the *Z-Offset* group to define the selected color channel as a reference for the Z-Offset.
 - This icon is now displayed on the selected color channel in the experiment plan.
 - The current Z-position is displayed next to the *Define Z Reference* button. This Z-position is used as a reference value for the Z-Offset of the other color channels.
 - As you can't enter a Z-Offset for the reference image, the *Z-Offset in μm* field is not available as long as the reference color channel is selected.



In this experiment plan a Z-Offset for the individual color channels is defined. The green color channel is used as a reference.

Select one of the color channels to view its Z-Offset values in the *Experiment Manager* tool window (1). In the example shown, the microscope's Z-drive is raised by 8 μm in relation to the current Z-position of the reference color channel before the acquisition of the red color channel.




- Select the next color channel within the multi-channel group.
- Choose the observation method that belongs to the selected color channel and focus the sample.
- Click the *Read Z-offset* button in the *Process Manager* tool window, to adopt the current Z-position of your microscope stage.



- Your software computes the difference to the reference color channel's Z-position and enters this value in the *Z-Offset in μm* field.
9. Define the Z-Offset for the other color channels within the multi-channel group.
- When you later load and run the experiment plan, focus before the acquisition of the reference color channel. The focus position of the remaining color channels is then adjusted accordingly.

Acquiring a multi-channel fluorescence image together with a transmitted light image

Expand your experiment plan and acquire a transmitted light image as well, with the brightfield observation method, for example.

1. Select the multi-channel group in the experiment plan. Use the mouse to enlarge the frame so that there's enough room inside the group for another image acquisition command.
2.  Click the small arrow next to the *Image Acquisition* button to open a menu. Choose an observation method for the acquisition of a transmitted light image, e.g., phase contrast, differential interference contrast (DIC), or brightfield.
Position the command to the right of the last fluorescence image acquisition inside the multi-channel group.
3. Connect the last fluorescence image acquisition command with the transmitted light image acquisition command. To do this, click on a control point and, while holding the left mouse button pressed, move the mouse pointer to the other control point.
4. Select the command for the transmitted light acquisition in the experiment plan.
 - In the *Experiment Manager* tool window, the *Image Acquisition*, *Camera Settings*, and *Display* groups are now shown.
 - The *Transmission overlay* check box in the *Image Acquisition* group is available when the command selected in the experiment plan is linked to a transmitted light observation method.
5. Select the *Transmission overlay* check box in the *Experiment Manager* tool window. Now the transmitted light image is assigned its own image layer on top of the fluorescence images.
 - 6.  This icon appears in the experiment plan on the command for the acquisition of the transmitted light image.
6. Make the rest of the settings for the acquisition of the transmitted light image.
7.  Click the *Start* button, located in the *Experiment Manager* tool window, to run the experiment.
 - Then, together with your fluorescence images, a transmitted light image will also be acquired and saved together with the multi-channel fluorescence image. The result of this acquisition process is a multi-layer image that you can view with the *Layers* tool window.



The illustration shows an experiment plan for a multi-channel acquisition with a transmitted light image (1).

The transmitted light image acquisition command is inside the multi-channel group. The experiment will produce a multi-layer image with two image layers. One image layer is the multi-channel image and the second layer is the transmitted light image.



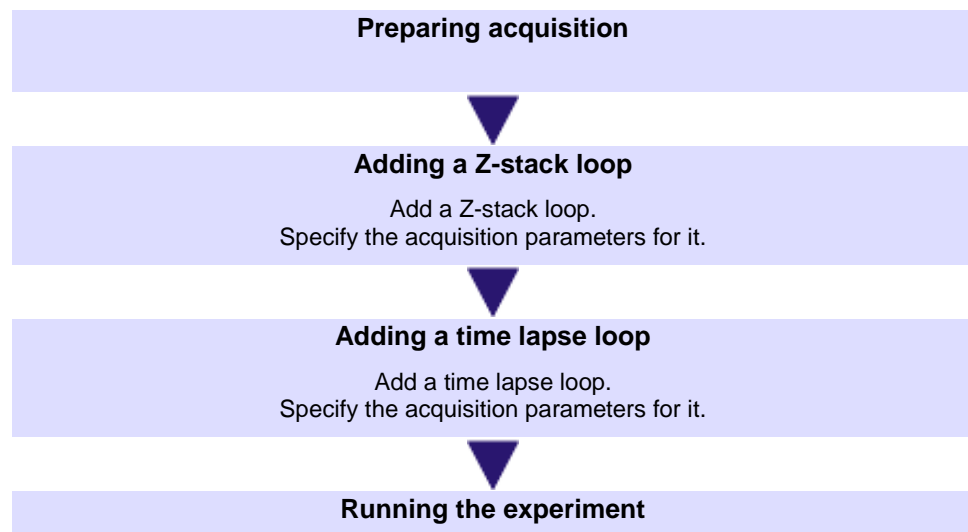
Alternatively, you can also use this experiment plan to acquire a multi-channel image with a transmitted light image. The transmitted light image acquisition command is, in this case, outside the multi-channel group. The experiment then produces two images, one multi-channel image and the transmitted light image. In this case, you can't place one images on top of the other to view the superimposition.

Acquiring multi-dimensional images

Task Define an experiment plan for acquiring a multi-channel Z-stack. You want the acquisition of the multi-channel Z-stack image to be repeated several times at intervals of one hour.
Run the experiment.

- Preconditions**
- The system has been configured.
 - You have defined suitable observation methods for your color channels.
 - Your microscope has a motorized Z-drive. The Z-drive has been set up and calibrated.

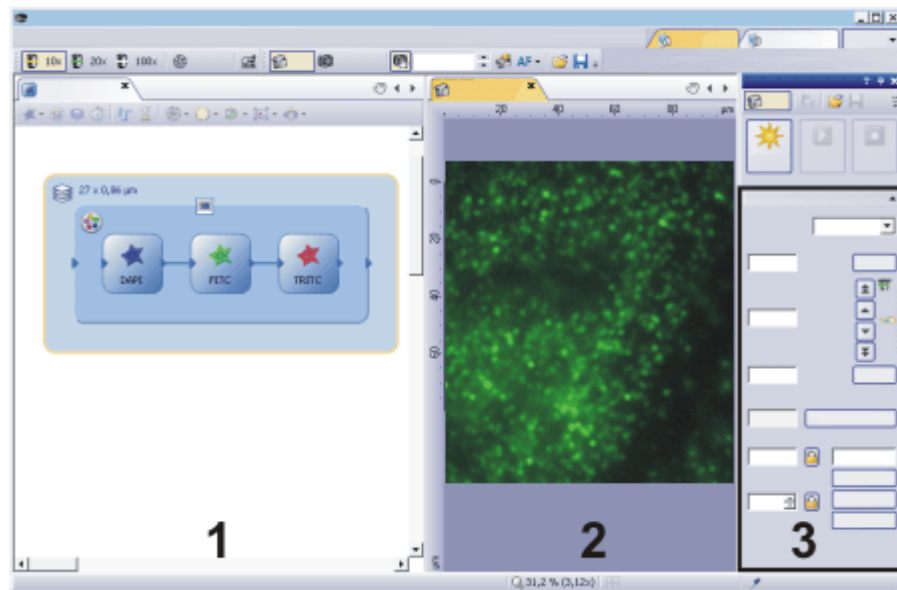
The following process flow chart displays the basic steps of the process.



1. Load an experiment plan for acquiring a multi-channel fluorescence image. Or specify a new experiment plan.
2. Use the *File > Save As...* command to save the experiment plan with a new file name.
3. Select one of the commands in the experiment plan, DAPI, for example. Click the *Get Settings* button in the *Experiment Manager* tool window to have the DAPI observation method set on the microscope.
4. Click the *Live* button, located in the toolbar at the top of the *Experiment Manager* tool window.
5. Arrange the live window and the experiment plan in the document window so that you can see both documents at the same time. To do this, use the *Window > Split/Unsplit > Document Group (Right)* command, for example.
6. Bring the image into focus.

Preparing acquisition





Some of the settings for the Z-stack acquisition can best be checked in the live-image. You can display the live-image (2) and the experiment plan (1) in the document window at the same time.

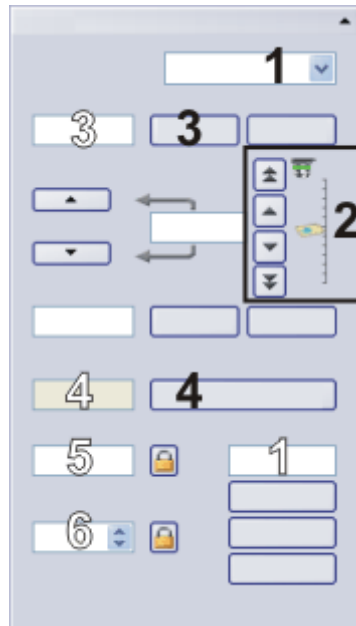
The Z-stack loop has been selected in the experiment plan. In the *Experiment Manager* tool window, the Z-stack group (3) is now visible. Set the acquisition parameters here.

Adding a Z-stack loop



7. Click the *Z-stack loop* button. You can find the button on the toolbar at the top of the experiment plan.
8. Draw a frame around the multi-channel group. Your microscope's Z-drive will now be automatically moved to a different Z-position when all the commands in the Z-stack loop have been carried out.
 - The *Z-stack* group is displayed in the *Experiment Manager* tool window. Set the acquisition parameters for the Z-stack loop here.
9. Define the acquisition parameters for the Z-stack loop in the *Experiment Manager* tool window.
 - In the case being described, first the whole multi-channel image is acquired at a Z-position. Only then does the stage move to the next position.
You can also define the experiment to acquire the whole Z-stack for one color channel first. Then the observation method is changed and then the whole Z-stack is acquired for the next color channel.

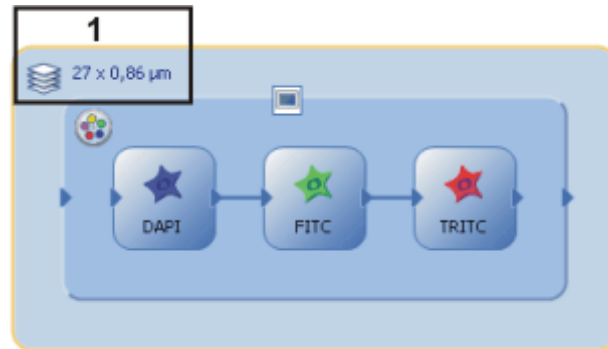
Selecting the acquisition parameters for the Z-stack loop



Set the acquisition parameters for the acquisition of a Z-stack in the *Experiment Manager* tool window. The acquisition parameters apply to the Z-stack loop that is selected in the current experiment plan. Use the fields and buttons with black numbers (1-4) for this. The values in the fields with white numbers are automatically calculated and updated by your software.


10. Select the *Top and bottom* entry in the *Define* list (1).
 - The stage's current Z-position will be shown in the *Position* (1) field. Because you've already focused, this is the focus position.
11. Using the arrow buttons (2), move the microscope's Z-drive up to the Z-position at which the structures that are directly under the surface of the sample are displayed in sharp focus. The buttons with a double arrow move the stage in larger steps. Now, move the Z-drive the same distance again in the same direction. Click the upper *Set* button (3).
 - The current Z-position will be adopted in the *Start* field (3).
12. Now, define the final position in exactly the same way.
 - The *Recommended Step Size* field (4) displays the distance required between two Z-positions in the Z-stack. The required distance depends on the objective's Numerical Aperture, among other things, and is automatically calculated using the Nyquist theorem. This makes sure that no positions on the sample between two individual images remain out of focus. The higher the objective magnification and your objective's Numerical Aperture is, the smaller the required distance will be.
13. If necessary, release both of the buttons showing the lock icon.
14. Click the *Apply* button (4) located next to the *Recommended Step Size* field.
 - The *Step Size* field (5) adopts the value from the *Recommended Step Size* field.
 - The *Z-Slices* field (6) now displays how many Z-positions will be moved to by the Z-stack loop. The number of Z-positions will be automatically calculated on the basis of the Start and End values, and the Z-spacing.

15. Finish the live mode.



The illustration shows an experiment plan for the acquisition of a multi-channel Z-stack image.

The experiment plan displays the number of Z-positions that will be moved to and the Z-spacing (1). In this example, 27 individual images with a Z-spacing of 0.86 µm will be acquired for each channel.

Adding a time lapse loop


16. Click the *Time Lapse Loop* button. You can find the button on the toolbar at the top of the experiment plan.

17. Draw a frame around the whole multi-channel Z-stack image. The acquisition of the multi-channel Z-stack image will now be repeated. All acquired images will be combined into a single multi-dimensional image, a multi-channel Z-stack image.

- The *Time Lapse Loop* group is displayed in the *Experiment Manager* tool window. Set the acquisition parameters for the time lapse loop here.

18. Define the acquisition parameters for the Z-stack loop in the *Experiment Manager* tool window.

Selecting the acquisition parameters for the time lapse loop

Set the acquisition parameters for the acquisition of a time lapse loop in the *Experiment Manager* tool window. The acquisition parameters apply to the time lapse loop that is selected in the current experiment plan. Use the fields with black numbers and the check box (1-3) for this. The values in the fields with white numbers are automatically calculated and updated by your software.

19. In the *Cycles* field (1), enter how often the command should be repeated in the time lapse loop. Enter the value 5 to acquire five multi-channel Z-stack images, for example.

20. Clear the *As fast as possible* check box (2).

21. In the *Interval* field (3), enter the time interval you want between two cycles. Enter the value 0,5 h, for example. Now the acquisition of a new multi-channel Z-stack image will begin half an hour after the start of the previous acquisition.

- The *Approximate minimum interval* field (1) displays the minimal time needed to carry out all the commands in the current time lapse loop. This

is the amount of time that will be needed when you select the *As fast as possible* check box.

- The *Total loop duration* field (2) displays the time needed to carry out the time lapse loop.

Running the experiment



The illustration shows a completed experiment plan for the acquisition of a multi-channel image. It consists of three nested loops. There is a multi-channel group (1), a Z-stack loop (2) and a time lapse loop (3). The result is an image file.



22. Click the *Start* button, located in the *Experiment Manager* tool window, to run the experiment.

- The experiment starts immediately. Nested loops are carried out starting on the inside and working to the outside. To be exact, in this case that means:
First a multi-channel fluorescence image is acquired.
After this has been done, the stage's Z-position changes, and another multi-channel fluorescence image is acquired at the new Z-position.
The Z-position keeps changing until all Z-positions have been used.
Your system waits for half an hour and then repeats the image acquisitions.



- The acquisition has been completed when you can once more see the *Start* button in the *Process Manager* tool window, and the progress bar has been faded out.
- Your system's hardware components are now set as specified for the last observation method that was used.
- The experiment produces a single multi-dimensional image.

Acquiring fast fluorescence time stacks

You can use the Experiment Manager to acquire very fast fluorescence time stacks. In this way, you can acquire kinetic processes with your system's highest possible time resolution.



1. Define an experiment plan with a fluorescence image and a time lapse loop.
2. Select the command for acquiring fluorescence images in the experiment plan.
Select the *Streaming* check box in the *Experiment Manager* tool window.
 - Pay attention to the experiment's total duration, shown at the top of the *Experiment Manager* tool window. The total duration display gets smaller which directly shows the effect the streaming is having.
 - The command for the fluorescence image acquisition in the experiment plan now looks slightly different.



On the left, you can see the command for a fluorescence image acquisition using the DAPI observation method.

On the right, the *Streaming* check box is selected. The status of the check box is indicated by a slightly changed icon in the experiment plan.

3. Select the command for the time lapse loop in the experiment plan.
Select the *As fast as possible* check box in the *Experiment Manager* tool window.
4. Click the *Start* button, located in the *Experiment Manager* tool window, to run the experiment.
 - All images in the time stack are now acquired at the maximum possible frame rate without waiting to be triggered by the software or the hardware. For example, you can specify an observation method that opens a shutter before the image acquisition and closes it again after the image acquisition. When you are using streaming, the shutter stays open for the whole duration of the time stack acquisition. Without streaming, the shutter would be opened before a single image acquisition and closed again.



This is how an experiment plan for the fast acquisition of a fluorescence time stack with the *TRITC* observation method looks.

Acquiring multi-channel fluorescence images at different positions on the sample


Task Define an experiment plan with which you can acquire multi-channel fluorescence images at different positions on the sample. The experiment should always start at a specific stage position.
Your sample has been stained with the DAPI and FITC and fluorochromes.

Preconditions

- The system has been configured.
- You have defined suitable observation methods for your color channels.
- Your microscope has a motorized Z-drive. The Z-drive has been set up and calibrated.
- Your microscope has a motorized XY-stage. The XY-stage has been set up and calibrated.

Adding stage positions



1. Load an experiment plan that acquires a multi-channel fluorescence image or create a new experiment plan.
2. Check the settings for each image acquisition command.
3. Click the *Stage Loop* button. You can find the button on the toolbar at the top of the experiment plan.
4. Draw a frame around the multi-channel group.
 - The *Stage Loop* group is displayed in the *Experiment Manager* tool window. Use the functions in this group to define the positions on your sample where a multi-channel fluorescence image is to be acquired.
5. Move the stage to the first position and focus.
 - To move the stage, you can use the joystick or the navigation wheel in the *Microscope Control* tool window. If you've acquired an overview image of your sample, you can also click on the position on the sample in the overview image in the *Stage Navigator* tool window.
6. Click this button in the *Stage Loop* group to add the current stage position to the position list.
 
7. Select more positions for the acquisition of the multi-channel fluorescence image.
 - The current number of positions is shown in the experiment plan.
 - When the experiment starts, the stage will go to all positions in the position list one after the other. The experiment will be carried out at each position that is defined inside the stage loop. In this example, a multi-channel fluorescence image will be acquired at each position.



The experiment plan contains a stage loop with 8 stage positions (1). A 2-channel fluorescence image is acquired at each stage position.

Checking stage positions

You can go back to stage positions that have already been defined to check them and, if necessary, to delete them from the position list at any time.



8. Select the stage loop in the experiment plan.
9. Click this button in the *Experiment Manager* tool window.
 - The *Position List* dialog box opens. It displays all currently defined stage positions.
10. Select an position in the list.
11. Click the *Go to* button to move the microscope stage to the selected position.
12. Click the *Delete* button to delete the selected position. Now, you can no longer use this position for the experiment.

Starting the experiment



13. Click the *Start* button, located in the *Experiment Manager* tool window, to run the experiment.
 - The stage moves to all stage positions one after the other. A multi-channel fluorescence image is acquired at each stage position.
 - The experiment will produce 8 multi-channel fluorescence images.

12.3.2. Using image analysis functions in an experiment

Measuring intensity profiles on a time-stack

Task

Acquire a fluorescence image and measure the intensity profile of 2 image segments on this image.

Preconditions

- The system has been configured.
- You have define observation methods for your color channels, FURA340 and FURA380 for example.

Defining ROIs



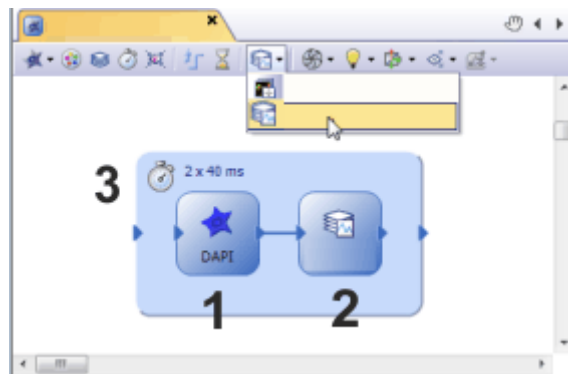
1. Use the *View > Toolbars > Life Science Application* command, to have the *Life Science Application* toolbar displayed.
2. Select a fluorescence observation method and acquire a fluorescence image.
3. Define two Regions of Interest (ROIs) on typical structures in the image. You can use one of the 3 ROI buttons on the *Life Science Applications* toolbar for this.
4. Use the *File > Save As...* command to save the image together with the ROIs. This step is optional. You can also carry out the experiment without saving the reference image.
 - This image will be used as the reference image for the ROIs in the next analysis.

Defining the experiment plan



5. In the *Experiment Manager* tool window, click the *New* button to create a new experiment.
6. Add the command for the acquisition of the fluorescence image to the experiment plan.
7. Click the *Intensity Profile* button. If the button isn't shown, click the small arrow next to the *Ratio* button and select the command from the menu. Place the intensity profile after the image acquisition.
8. Draw a frame for a time lapse loop around both of the commands (image acquisition and intensity profile).





The illustration shows an experiment plan for the computation of an intensity profile. The experiment plan contains commands for image acquisition (1), for the computation of the intensity profile (2), and a time lapse loop (3).

Selecting the acquisition parameters

9. Select the icon for the image acquisition in the experiment plan. Now make all the settings for the acquisition of the fluorescence image in the *Experiment Manager* tool window. Take special care to select a suitable exposure time.
10. Make all of the settings for the acquisition of the time lapse loop. Select the *As fast as possible* check box to acquire the images at the shortest possible intervals.

Define Intensity Profile

11. Select the intensity profile in the experiment plan.
 - The *Intensity Profile* group is displayed in the *Experiment Manager* tool window. Make the settings for the computation of the intensity profile here.
12. Select the image on which you defined the ROIs from the *Reference Image for ROIs* list.
 - If you haven't defined any ROIs on a reference image, the intensity profile is automatically computed for the whole image.
13. Select the *Compute Average > For Each ROI* option.
14. Select the *Auto* entry from the *X-axis in diagram* list.

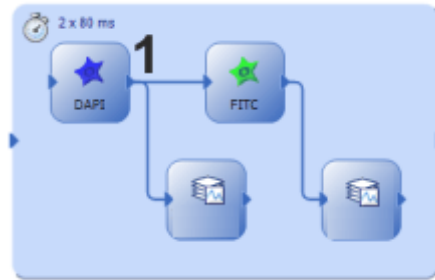
Starting the experiment



15. Click the *Start* button, located in the *Experiment Manager* tool window, to run the experiment.
 - The experiment starts immediately. Nested loops are carried out starting on the inside and working to the outside. To be exact, in this case that means:
First, several fluorescence images are acquired one after the other and are combined into a time stack.
Next, the intensity profile is computed for all the ROIs that were defined on the reference image.
 - The *Intensity Profile* tool window appears under the experiment plan and you can look at the intensity profiles of the ROIs.

Expanding the experiment plan

16. Expand the experiment plan. Inside the time lapse loop, add a command for the acquisition of another fluorescence image. Connect the control points for both image acquisition commands with each other.
17. Drag the command for the computation of the intensity profile to the bottom of the experiment plan.
 - There are now two connectors at the image acquisition command's output control point.



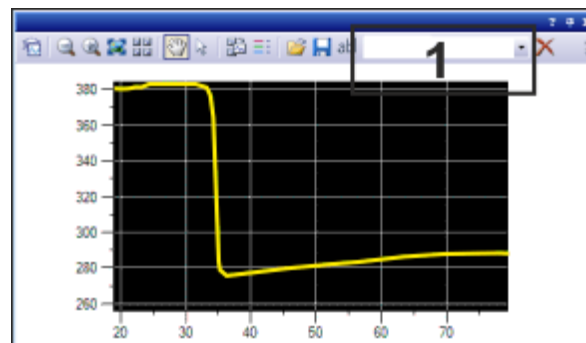
The *Intensity Profile* command can branch out from the connector to other commands in order to keep a clear overview of the experiment plan.

18. Have the intensity profile computed on the second image as well.

- This experiment now produces time stacks for different fluorescences at the same position on the sample.
- You can view the computed intensity profiles in the *Intensity Profile* tool window. In the list in the tool window's toolbar, select the image whose intensity profile you want to view.



19. Click the *Start* button, located in the *Experiment Manager* tool window, to run the experiment.



After the experiment is finished, you can view the computed intensity profiles in the *Intensity Profile* tool window. If you have computed intensity profiles for more than one image, you can find a separate record for each image in the list (1).

Carrying out a Ratio Analysis

Task A typical application for ratio analysis is the measurement of the concentration of free calcium ions. Calcium ions are released by extracellular signals like a nerve impulse, for example. The Fura-2 fluorescence dye is used for the measurement of ion concentrations because its excitation level shifts from 380 nm to 340 nm with an increasing ion concentration. A time stack for both the Fura340 and Fura380 fluorescence channels is acquired during a change in the concentration of calcium ions. For the ratio analysis, the images for both of the channels are divided by each other to make the changes in intensity clearly visible. Use the online ratio analysis to compute the ratio image over time and the relation of the intensity profiles in two cells.

Preconditions

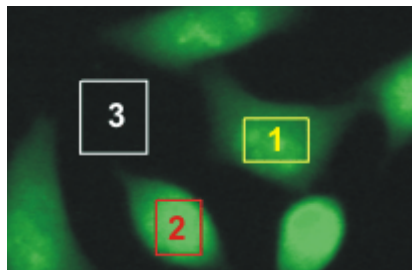
- The system has been configured.
- You have defined suitable observation methods for your color channels.

Defining ROIs

1. Use the *View > Toolbars > Life Science Application* command, to have the *Life Science Application* toolbar displayed.
2. Select a fluorescence observation method, the *Fura380* observation method, for example. Acquire a fluorescence image.
3. Define 2 ROIs in different cells and a third ROI on the dark background for each image.



- You can use one of the 3 ROI buttons on the *Life Science Applications* toolbar for this.
- This image will be used as the reference image for the ROIs in the next analysis.

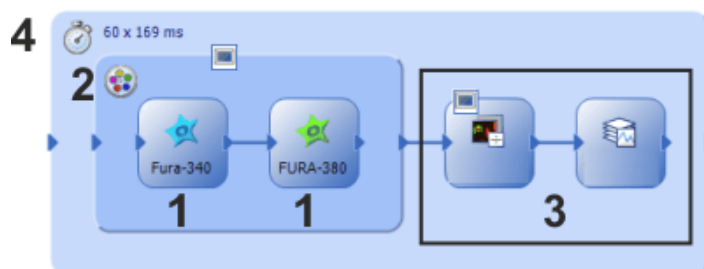


Three ROIs have been defined on the image. The red and yellow ROIs contain cells. The white ROI is on the background. You need ROI (3) during the ratio analysis for the calculation of the background. The intensity profiles are calculated on ROIs (1) and (2).

Defining the experiment plan



4. In the *Experiment Manager* tool window, click the *New* button to create a new experiment.
5. Define the following experiment plan.



The experiment plan contains two fluorescence channels (1) in a multichannel group (2) and the commands for the computation of a ratio image and an intensity profile (3), all within one time lapse loop (4).

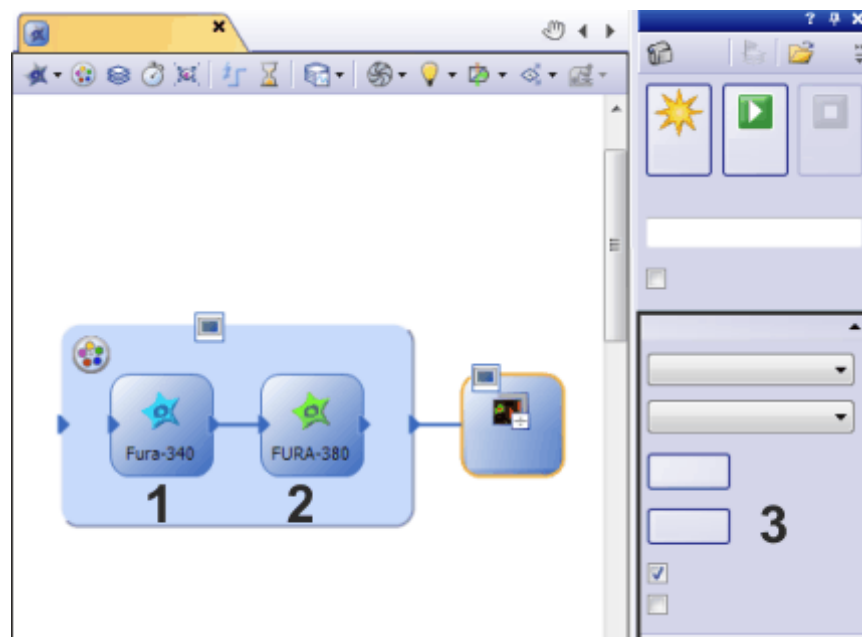
Defining the acquisition of fluorescence images and time lapse loops



Defining the ratio analysis

6. Make all of the settings for the acquisition of the fluorescence channels and the time lapse loop. Select the *As fast as possible* check box for the time lapse loop to acquire the images at the shortest possible intervals.
7. Select the command for the ratio analysis in the experiment plan.
 - The *Ratio* group is displayed in the *Experiment Manager* tool window. Make the settings for the measurement here.
8. Select the *Fura340* channel from the *Numerator* list and the *Fura380* channel from the *Denominator* list. The ratio analysis divides the Fura340 image by the Fura380 image.

Note: In the *Experiment Manager* tool window, the *Numerator* and *Denominator* lists don't contain the names of the current observation methods, but the general names *Channel 1*, *Channel 2* and so on. The general names always refer to the color channels that are in the current experiment plan in the multi-channel group before the *Ratio* command.



In the illustration, the *Ratio* command is selected in the experiment plan. That's why the *Ratio* group (3) now appears in the *Experiment Manager* tool window. The *Ratio* command has to be positioned after a multi-channel group. The first color channel in the multi-channel group (1) is referred to as *Channel 1* in the *Numerator* and *Denominator* lists. The second color channel (2) is *Channel 2*.

9. Click the *Options...* button.
 - The *Ratio Analysis* dialog box opens.
10. In the *Ratio Analysis* dialog box, select the following settings. In both *Thresholds* lists, set a value of 5 to suppress image noise. In the *Scale* list, adopt the value of 1000. Select the *ROI* option in the *Background* group. Select the image on which you defined the ROIs and the names of the ROIs that you defined on the background from the *Numerator* and *Denominator* lists.
11. Close the *Ratio Analysis* dialog box with *OK*.
12. The ratio analysis produces a ratio image. Select the *Store result* check box if you want to save the image resulting from the ratio analysis.
 - The ratio image is now saved together with the acquired multi-channel time stack. The resulting image is a multi-layer image in which one image layer is the ratio image and the other image layer is the multi-channel time stack.

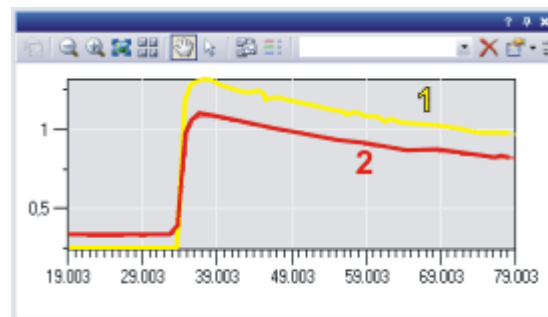
- You can specify the storage location for the resulting image in the *Acquisition Settings > Saving > Process/Experiment* dialog box. You can find more information on this dialog box in the online help.

Define Intensity Profile

- Select the intensity profile in the experiment plan.
- In the *Experiment Manager* tool window, make the following settings in the *Intensity Profile* group.
 - Select the *Compute Average > For Each ROI* option.
 - Select the *Auto* entry from the *X-axis in diagram* list.
 - Select the image on which you defined the ROIs from the *Reference Image for ROIs* list.

Starting the experiment

- Click the *Start* button, located in the *Experiment Manager* tool window, to run the experiment.
- If you are using Fura to measure the change in the calcium ion concentration, add a signal molecule to the cell culture.
 - The *Intensity Profile* tool window appears under the experiment plan and you can look at the intensity profiles of the ROIs.



The intensity profiles show how the ratio value in both ROIs (1) and (2) changes over time. The colors of the intensity profiles correspond to the colors of ROIs they describe.

12.3.3. Adapting existing experiments

You can view and edit the settings for individual commands at any time.

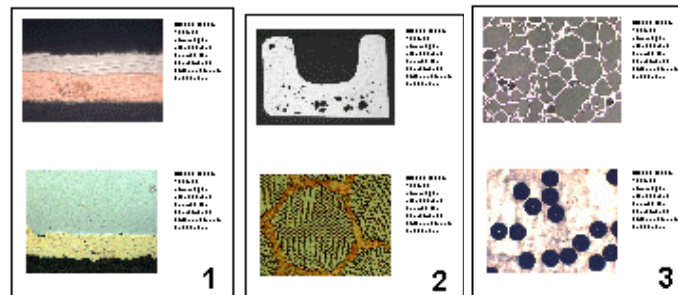
1. Should the *Experiment Manager* tool window be hidden, use the *View > Tool Windows > Experiment Manager* command to make it appear.
2. Load a saved experiment plan. Experiment plans have the OEX file name extension.
3. Select the command in the experiment plan that you want to edit, an image acquisition for example.
 - You can view and change all current settings for the selected command in the *Experiment Manager* tool window.
4. Save the experiment plan with the changed settings.

00801 15072014

13. Working with reports

The report functionality enables you to document the results of your work and to make them available to third parties. You can publish reports as a DOC-file or DOCX-file (for MS-Word 2007/2010) or as a printout. When a corresponding printer driver is available on your system, you can also print reports to a PDF, and send them to other users.

The illustration shows a report example that is made up of three pages, and contains six images.

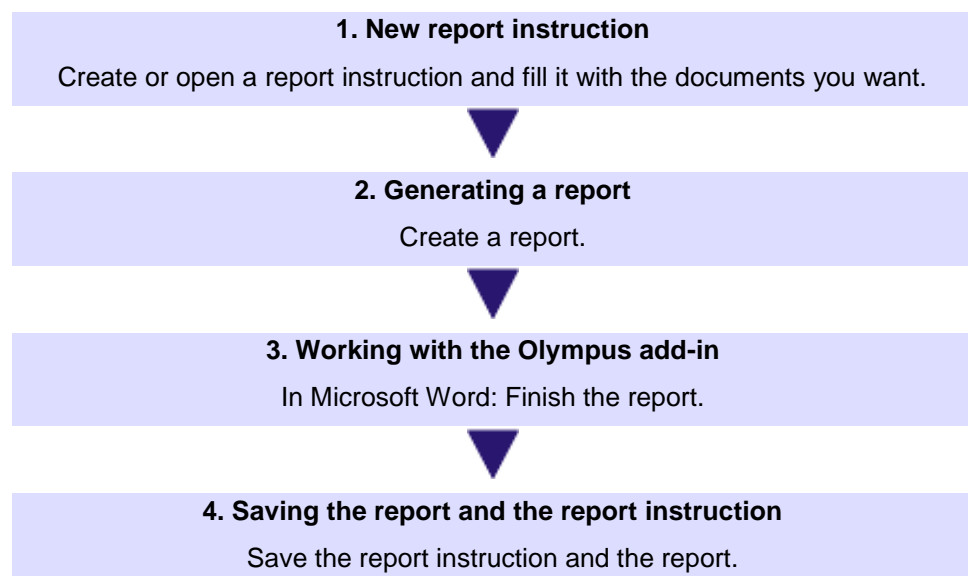


When working with reports, you have to switch to the *Report* layout. In this layout, the *Report Composer* tool window, that is required when working with reports, is by default displayed.

The first step when working with reports is always opening or creating a new report instruction in your software. In the report instruction, you specify which images and which page layout are to be taken for the report. The display and any further editing of the created report are done in the MS-Word application program. Therefore, this program has to be installed on your PC when you work with reports.

Note: Two programs are involved in the creation of reports: Your software and the MS-Word application program. You can use MS-Word 2003, 2007 or 2010 for working with reports.

Process flow when you create a report



13.1. Working with the report composer

The *Report Composer* tool window supports you when you are creating and updating report instructions. In this tool window, you also find the *Create* button that is used to start the report creation.

Note: Two programs are involved in the creation of reports: Your software and the MS-Word application program.

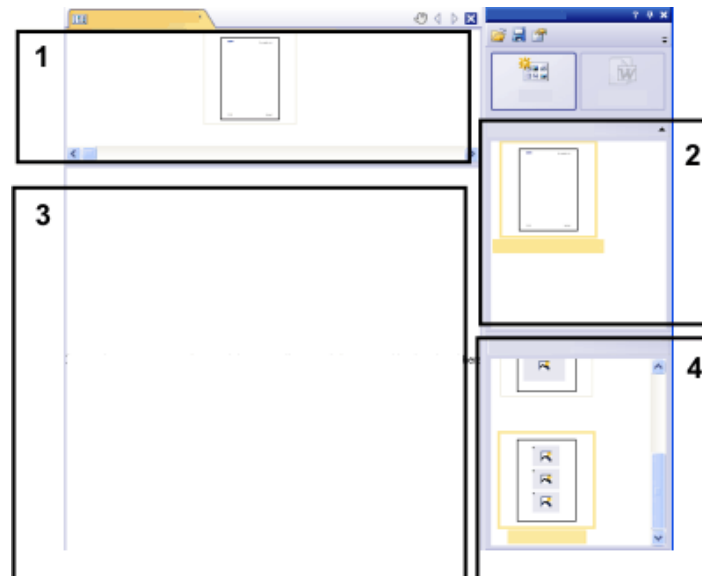
Should the *Report Composer* tool window be hidden, use the *View > Tool Windows > Report Composer* command to make it appear.

Creating a new report instruction

To create a report, first create a new report instruction in your software. You can also use a saved report instruction.

Note: The report instruction has to contain at least one registered page template. You can find more information on registering page templates in the online help.

1. Switch to the *Reporting* layout.
2. Click the *New Report Instruction* button. You find this button in the *Report Composer* tool window.
 - A new document of the "report instruction" type will be created in the document group. This document is at the same time the workspace in which you put the report together.



3. If no default document template has been defined: Drag the document template you want onto the upper part (1) of the report instruction. You find a list of the available document templates in the upper part (2) of the *Report Composer* tool window.
 - If a default document template has been defined, it will be automatically inserted in the upper part of the new report instruction.
 - Creating a report is also possible when you leave the upper part of the report instruction empty. In this case, the default MS-Word document template is used.
4. Drag the page templates you want onto the lower part of the report instruction (3). You find a list of the available page templates in the lower part (4) of the *Report Composer* tool window.
 - Every report has to contain at least one page template.

- Make sure that the page templates contain the correct placeholders for the document types that you want to drag onto the report instruction. Accordingly, if your report is to contain an image and a chart, select a page template that contains one placeholder for an image and another for a chart.
- If you want to use workbooks in your reports, MS-Excel must be installed on your PC. The minimum MS-Excel version required is MS-Excel 2003.
- The placeholder for a workbook can also be used for a MS-Excel file. To do so, select the MS-Excel file in the *File Explorer* tool window and drag it onto the report instruction. In the report instruction, MS-Excel files are shown with this icon:

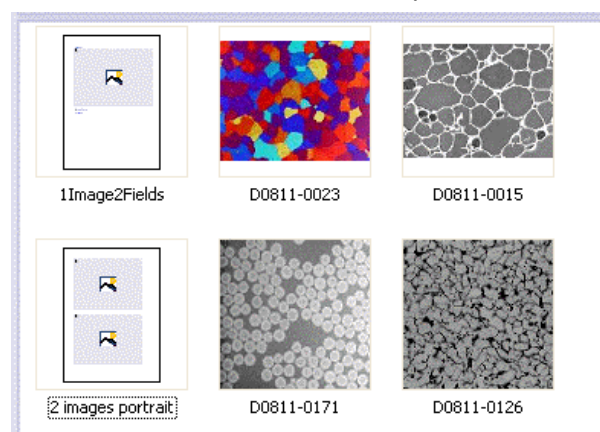


5. Drag the documents you want onto the lower part of the report instruction (3).

- In the *Reporting* layout, the *Database*, *Gallery* and *File Explorer* tool windows are arranged to the left of the document window. In each of the tool windows you can select one or more documents and drag them onto the report instruction. If you use the *File Explorer* tool window, the documents do not need to be open for this. If you use the *Database* tool window, the documents don't have to be open either. It is sufficient to open the database. However, the *Gallery* tool window only allows you to select documents that are currently open in your software.
- You can also integrate MS-Word files (e.g., background information regarding the project) into your MS-Word reports. MS-Word files don't need a placeholder in the report instruction. Select the MS-Word file in the *File Explorer* tool window and drag it directly onto the report instruction. In the report instruction, MS-Word files are shown with this icon:



- The documents must have been saved, because unsaved documents cannot be included in a report.



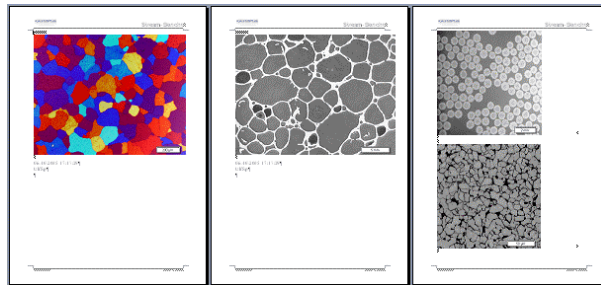
The illustration shows an example of a report instruction. In the report, two different page templates are to be used. The first page template contains a single placeholder for an image, the second page template contains two placeholders for an image. After the page template, the images that are to be inserted in the report page are displayed.

6. Check the report instruction now. You may still edit it and, e.g., delete or shift documents or select another page template.

Generate Report



1. Click the *Create* button. You find this button in the *Report Composer* tool window.
 - The report will be created. Creating a report can take some time when large reports with many images and documents are involved. Pay attention to the progress bar that is shown. The MS-Word application program will open automatically and display the new report. In the example shown below, the report has three pages. (The fact that the first page template only contains one image placeholder and two images have been selected in the report instruction, automatically leads to the creation of two report pages.)



2. If you want to, you can still make additional changes in the MS-Word application program. To do so, use the Olympus add-in or the *Olympus* toolbar.
3. If you want to, save the report instruction and the report.

Editing a report instruction

You can make the changes described below to a report instruction. These changes do not apply to reports that have already been created on the basis of this report instruction. Therefore you must create a new report in order to see the changes you made. This will generate a new MS-Word document. Any changes that you may have made in the first version of the report will not be contained in the newly created DOC-file.

Exchanging the document template

1. Load the report instruction that you want to edit.
 - Report instructions have the file extension RCI.
2. To delete a document template, select it and press the [Del]-key on your keyboard.
3. Drag the new document template onto the upper part of the report instruction.
 - By doing so, the document template is exchanged. Please note that a report instruction can only contain one document template.
 - A report instruction must not contain a document template at all. When you leave the upper part of the report instruction empty, the MS-Word default document template will be taken.

Changing the page templates

1. Load the report instruction that you want to edit.
2. In the report instruction, select the page template you want to exchange.
3. Use the [Del] key on your keyboard to delete the selected page template from the report instruction.
 - By doing so, you only deselect the page template, no file will be deleted.
4. Drag the new page template to the position in the report instruction, where the deleted page template had been located.
 - Every report has to contain at least one page template.

Shifting the page templates

1. To shift a page template to another place in the report instruction, select it and, with the left mouse button depressed, drag it to a new position (Drag&Drop).
 - In certain cases, this may change the appearance of the report considerably. All documents that come after this page template in the report instruction will use this page template in the report.

Deleting documents

1. Load the report instruction that you want to edit.
2. In the report instruction, select the documents that you want to delete.
 - The standard MS-Windows conventions apply to the multiple selection.
3. Use the [Del] key on your keyboard to delete all of the selected documents in the report instruction.
 - By doing so, you only undo the document selection, no file will be deleted.

Adding documents

- You can add new documents to an existing report instruction at any time.
1. Load the report instruction that you want to edit.
 2. Simply drag the new documents onto the position you want in the report instruction.

- Dragging & dropping images onto the report instruction is possible from the *Database*, *Gallery* and *File Explorer* tool windows.
- Please note the page templates must be placed before the images.

Moving documents

You can change the order in which the selected documents are arranged in the report instruction at any time.

1. Load the report instruction that you want to edit.
2. Select an image, and with the left mouse button depressed, drag it to another position (Drag&Drop).

00153 30012012

13.2. Working with the Olympus MS-Word add-in

When your software is installed, an add-in from Olympus is added to the MS-Word application program. When you start MS-Word, you can recognize this because the *Olympus* menu is displayed.

Depending on the MS-Word version used, the Olympus MS-Word add-in looks slightly differently. In MS-Word 2003, the *Olympus* menu and the *Olympus* toolbar are available. In MS-Word 2007 and MS-Word 2010, only the *Olympus* menu is available. When it is selected, the commands are available in a ribbon.

This add-in will help you to carry out important tasks that are necessary when working with reports. It will support you with tasks that prepare the report creation as well as tasks that are for editing the already created reports.

Preparatory tasks

1. You define page templates that you want to use for your reports. For each page template you define placeholders for the document types you want.

Note: During the installation of your software some predefined page templates have been installed, too. Please check first whether these predefined page templates are already sufficient for your needs before creating your own page templates.

2. In case you did not insert the placeholders in a chronological order: You adjust the insertion order of the placeholders, so that they are numbered consecutively, from the first placeholder to the last one.

Tasks for editing the already created reports in MS-Word

1. You add a document that is currently open in your software, to a report (or to any other MS-Word file).
2. You add a field to your report, so that certain information that is saved in your software, is also displayed in MS-Word. This makes sense, for example, when, exceptionally, you want to see the acquisition date of a certain image in the report.
3. You change the image properties and set, for example, whether or not the info stamp and the scale bar should be shown.
4. You add one or more detail zooms to an image.
5. You change the resolution of one or all images of the report. If you want to share the report, it may be sensible to reduce the resolution, thereby also reducing the file size.

6. You update all placeholders in your report. This makes sense, for example, when you change the documents in your software after the report creation and when you now want to see them in the report.
7. You insert an MS-Word report into the database of your software. This command is only available if your software supports the database functionality.

Inserting documents and fields from the software in any MS-Word document you want

With the commands from the *Olympus* add-in you can also insert documents and fields from your software in any MS-Word document that hasn't been set up via the *Report Composer* tool window. In this case, you work mainly in the MS-Word application program and you switch to your software only when you want to open the documents that you want to insert into the MS-Word document.

10404 30012012

13.2.1. Creating and editing a page template

In a page template, placeholders are set up for the documents that the report is to contain. There are placeholders for images, charts, workbooks and fields.

When, for instance, the report is to contain pages that have an image at the top, and below it, a chart, you should then set up a template, which has a placeholder for an image and a placeholder for a chart.

The *Olympus > Templates* menu contains all of the commands that you need for setting up or changing a page template.

Creating a page template and adding a placeholder for a document

Note: Page templates have the DOC or DOCX (for MS-Word 2007/2010) file format. To be able to carry out the following step-by-step instructions, you must therefore have opened MS-Word, and have the *Olympus* menu on display.

1. In the MS-Word application program, select the *File > New...* command.
2. Select the *Empty Document* option, if you don't want to use an existing page template as template, but instead want to start from scratch.
3. Decide whether you want to insert a placeholder for an image, a chart, or a workbook. Select the *Olympus > Templates* command and choose one of the following options: *Insert Image Placeholder*, *Insert Chart Placeholder*, *Insert Workbook Placeholder*. You can find more information on workbooks in the online help.
 - The placeholder you've selected will be inserted.
4. If necessary, you can change the size of the placeholder. To do so, move your mouse over a handle, then with the left mouse button depressed, drag it in the required direction. The length/width ratio remains unchanged, so that the objects won't be distorted by this action.
5. Double click a placeholder for an image to change the default settings for its appearance.
 - The *Image properties* dialog box opens. You can find more information in the online help.
6. If required, insert additional placeholders for images, charts or workbooks.
 - Make sure that your page template isn't longer than a page. It's better to set up two page templates, each of one page, than it is to set up one page template that is made up of two pages.

7. If you want to, you can insert a placeholder for a field. Additional information about a placeholder can be shown in this field, for example, the name, or the date it was set up. You will find additional information on inserting placeholders for fields further down.
8. Save your page template in DOC or DOCX (for MS-Word 2007/2010) file format. You can choose any storage place you like.
9. If required, add a document title. In this case, your software shows this title in the *Report Composer* tool window. If you don't enter a document title, the file name is shown instead in the *Report Composer* tool window. To define a document title, do the following:
 - In MS-Word 2003: use the *File > Properties* command and switch to the *Summary* tab. Enter the document title in the *Title* field. Close the dialog box with *OK*.
 - In MS-Word 2007/2010: use the *File > Save as...* command. In the *Save as* dialog box, enter the document title in the *Title* field. Close the dialog box with *OK*.
10. Close the file.
11. Register the page template in your software. You can find more information on this topic in the online help.

Adjusting the insertion order

The placeholders are numbered in the order in which they were inserted. Should you have initially set up placeholders for two images, have then decided to put a placeholder for a chart right at the top of the page, the insertion order would be that shown in the example on the left.

1. In this case, use the *Olympus > Templates > Adjust Insertion Order* command to have the insertion order numbered serially from top to bottom (see example).



Inserting a placeholder for a field

1. In the page template, select the placeholder into which you want to insert a field.
2. Use the *Olympus > Templates > Insert Field Placeholder...* command.
 - The *Insert Field* dialog box opens. A description of this dialog box can be found in the online help.
3. Click the *Use Selected Placeholder* button.
 - In the *Placeholder* list, the name of the placeholder into which you want to insert a field appears.
4. In the *Available fields* list, select the field that is to be inserted. The entries in this list are arranged hierarchically. Click the plus sign to expand the list.
 - Two types of field are available.
 - The *Document Properties* list contains fields that are, by default, in your software, managed for this document type.
 - The *Database fields* list contains all of the fields that are available in the database for the selected placeholder. For this purpose, a database must have been opened.
5. In the *Insert Field* dialog box, click the *Insert* button.
 - The placeholder for a field will then be displayed. You can recognize it by the curly blue bracket, and by the field name shown.
6. If required, move the field to another position in the MS-Word file. By default, field values are shown to the left of, or above, the selected placeholder.
7. If necessary, add placeholders for further fields.
8. Save the page template.

00402 30012012

13.2.2. Editing a report

Preliminary considerations

When you have created a report and want to make some changes in it, before doing so, you should decide whether it will be better to make the changes in the report (i.e., in MS-Word) or in the report instruction (i.e., in your software).

Often, it is advisable to change the report instruction first and then create a new report. Changes you make in the report instruction are valid for every subsequent report that you create with this report instruction. There are numerous changes that you can anyway, only make in the report instruction, for example, the selection of other page templates.

Changes that you make in a report, are only valid for that particular report. There is also no possible way in which changes that are made in a report can be automatically adopted in the report instruction.

However, there are some cases when it makes sense to make a change only in the report, for example when you've created a very large report with a large number of documents, and only want to change the order of two images in it. If you didn't use a report instruction, you can only edit the report anyway.

Changing the image properties

When images are transferred to an MS-Word report, the image link is transferred as well. This makes it possible to change the image display in an MS-Word report (e.g., to scroll the image segment).

1. Doubleclick the placeholder for the image, to open the *Image Properties* dialog box.
2. In the *Display* group, select the *Scale bar if calibrated*, *Info Stamp* and *Border* check boxes, if these elements are to be displayed.
 - The properties of these elements can be defined in the *Options > Image Information* dialog box. Click the *Options...* button to open this dialog box.
3. In the *Size* group, select one of the options that specify how large the image is to be displayed in the report. You can find more information on these options in the online help.
4. If your settings should apply for all future images, click the *Set as Default* button.
5. Click *OK*.
 - The *Image Properties* dialog box closes. The changed image properties will be shown in the MS-Word report now.

Adjusting documents

In the MS-Word report, you can select a document of the "image" or "chart" type and select the *Olympus > Adjust Document* command. You will then change over to the image analysis software, where you can edit the document and then automatically change back to the report.

Note: Basically, the *Adjust Document* command is similar to the *Update Placeholders* command. The difference is that the *Adjust Document* command is meant for users who mainly use MS-Word for creating reports and who only change over to the image analysis software in order to make certain changes to some documents.

Example:

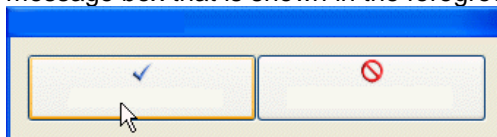
In the MS-Word application program, you edit a report that contains a lot of images. With a certain image you notice that an important measurement is missing. Using the *Adjust Document* command, you change over to the image analysis software, add the missing measurement and then change back to MS-Word in order to continue editing the report.

Adjusting an image

1. Open the report in MS-Word and select the image that you want to adjust.
2. Select the *Olympus > Adjust Document* command.
 - You switch to the image analysis software. If it was closed, it will be started and displayed in the foreground.
 - The image that you want to adjust is also opened. In case it is from a database that is currently closed, the database will be opened in the background.

Note: The image analysis software is now in a special "adjust-document" mode. In this mode, you can only make certain adjustments to the image. This is why a lot of other functions are hidden.

3. Make the required change.
4. If the image information was changed: Save the image in the image analysis software.
 - Some changes made to an image don't have to be saved, e.g., when you select another frame in a multi-dimensional image. Other changes have to be saved, e.g., adding a measurement. The fact that a change has to be saved will be indicated by an asterisk shown behind the file name in the document group.
5. Click the *Update Report* button. You find this button in the *Adjust Document* message box that is shown in the foreground.



- The MS-Word application program will now be shown in the foreground again. The edited image will be displayed. You can now continue to edit the report.
- If your image analysis software was closed before you selected the *Adjust Document* command, it is closed again. If any images or databases had to be opened for this command, they will be closed as well.

Editing a workbook in the report

Your software supports the handling of workbooks. A workbook is created, for example, when you open the *Measurement and ROI* tool window and export a results sheet.

You can find more information on workbooks in the online help.

Note: If you want to use workbooks in your reports, MS-Excel must be installed on your PC. The minimum MS-Excel version required is MS-Excel 2003.

Apart from the "image" and "chart" document type, reports can also contain workbooks. A workbook is imported as an Excel object in MS-Word. You can further edit it in the report.

1. In the report, double click on the workbook.
 - You will change into the edit mode. You can recognize it by the fact that now the column headers and the row numbers are shown. In edit mode, as well as that, you can see all of the workbook's worksheets.
2. If need be, select the worksheet that you want to edit.
3. Double click the workbook in order to switch to edit mode. Make the required change.
 - When you want to format individual cells differently, select the cell and use the *Format Cells...* command in the context menu.
 - When you want to format the complete worksheet differently, (e.g., other font or other background color), select the complete worksheet (e.g., with the keyboard shortcut [Ctrl + A]), then select the *Format Cells...* command in the context menu.
 - When you want to hide a column, click on the column's header, then select the *Hide* command in the context menu.
4. Exit edit mode by clicking on any point in the MS-Word report, outside the workbook.

Changing image resolution

By default, all images of a report are transferred to MS-Word reports with a resolution of 192 dpi. In certain cases, it can make sense to change the resolution of individual or all images in a report. For example, if you want to print the report, you can raise the resolution. Alternatively, if you want to publish the report on the Internet, you can reduce the resolution.

1. Open the report in MS-Word. Decide whether you want to change the resolution of all images or just certain images
2. If you only want to change the resolution of one individual image, select that image. If you want to change the resolution of all images, you don't have to select any.
3. Select the *Olympus > Change Image Resolution...* command.
 - The *Change Image Resolution* dialog box opens.
4. Select the option you want in the *Apply to* group. You can choose between *Selected images* and *All images in report*.
 - The *Selected images* option is inactive if no images were selected while opening the command.
5. Specify in the *Image Resolution* group how you want to change the image resolution. If you choose the *User-defined* option, you can enter any resolution of your choice between 96 and 600 dpi into the *DPI* field.
6. Click the *OK* button to change the image resolution.
7. Check whether you are satisfied with the changed image resolution. If not, change the image resolution anew.
 - You can first reduce the image resolution, then save the report and then increase the image resolution again. This is possible because each time that you open the *Change Image Resolution...* command, the image is transferred from your software to MS-Word.
8. Once you are satisfied with the changes to the image resolution, save the report. Take a look at the new file size in the Windows Explorer.

Updating placeholders

The *Update Placeholders...* command makes it easy to have any changes made to the images after the report has been created also shown in the report. Please take note that all changes made in your software have to be saved, if they are to be displayed when the *Update Placeholders...* command is used.

In MS-Word, you open a report that you created some time ago. In the meantime, you had changed a lot of images in your image analysis software (e.g., added measurements). Now, the report is to be updated so that it shows the newest version of all of the images.

1. If you only want to update one placeholder, select just that one.
2. In MS-Word, use the *Olympus > Update Placeholders...* menu command.
 - The *Update Placeholders* dialog box opens.
3. In the *Update Placeholders* dialog box, specify whether or not all placeholders should be updated.
4. Select the *Update fields linked with placeholder(s)* check box if your MS-Word report contains fields which should also be updated.
 - You can find more information on this topic in the online help.
5. Click *OK*.
 - The placeholders will be updated.

Insert Document

In a report (or in any other MS-Word document), you can insert a document at any position. When you have, for example, created a report, and while you are viewing it, notice that you've forgotten an image, you can retroactively insert it into the report.

1. Use the *Olympus > Insert Document...* command.
 - The *Insert Document* dialog box opens.
2. In the area on the left, select the source the document comes from. You have the following possibilities:
 - Select the *Open Documents* entry if you want to insert a document that is currently opened in your software.
 - Select the *Database* entry if you want to insert a document that is part of the currently selected database folder. For this purpose, the database must be opened in your software. Should you work with a version of the software that doesn't support databases, the *Database* entry is hidden.
 - Select the *File Explorer* entry if you want to insert a document that is stored on your PC or in your network.
3. Select the required document in the document preview. Click the *Insert* button.
 - The required document will be inserted into the MS-Word report.
 - The *Insert Document* dialog box remains open.
4. Insert further documents now or close the dialog box.
 - The path of all documents that you inserted will be saved. That enables you to later update the inserted documents by using the *Olympus > Update Placeholders* command (in case the documents were changed after they have been inserted into the report).

Inserting a field

You can insert a field into an MS-Word report that describes the image in more detail. All of the values that have been saved in your software for this document can be displayed in this field.

1. Select the image in the report to which you want to insert a field.
2. Select the *Olympus > Insert Field...* command.
 - The *Insert Field* dialog box opens.
3. Click the *Use Selected Placeholder* button.
 - In the *Placeholder* list, the name of the placeholder into which you want to insert a field appears.
4. In the *Available fields* list, select the field that is to be inserted. The entries in this list are arranged hierarchically. Click the plus sign to expand the list.
 - Two types of field are available.
 - The *Document Properties* list contains document-specific fields, just as they are shown in your software's *Properties* tool window.
 - The *Database fields* list contains all of the fields that are available in the database for the selected image. For this purpose, a database must have been opened.
4. Check at the bottom of the dialog box, whether a field value is displayed in the *Field value* group. Should the *Field value* group remain empty, this means that the field hasn't been filled out in your software. In this case, select another field that has been filled out, instead.
5. Click the *Insert* button.
 - The field contents will be displayed in the MS-Word document.
7. If required, move the field to another position in the MS-Word file. By default, field values are shown to the left of, or above, the selected placeholder.
8. If necessary, add further fields.
9. Close the *Insert Field* dialog box.
10. Save the MS-Word report.

Note: Should you want to have the contents of a specific field regularly shown in your reports, you can already insert this field, (that is to say a placeholder for this field) into the page template. Then this field will be automatically filled out in every report.

00403 17022012



Manufactured by

OLYMPUS CORPORATION

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

Distributed by

OLYMPUS EUROPA SE&CO.KG

Wendenstrasse 14-18, 20097 Hamburg, Germany

OLYMPUS SCIENTIFIC SOLUTIONS AMERICAS CORP.

48 Woerd Avenue Waltham, MA 02453, U.S.A.

OLYMPUS SINGAPORE PTE LTD.

491B River Valley Road, #12-01/04 Valley Point Office Tower, Singapore 248373

OLYMPUS AUSTRALIA PTY. LTD.

3 Acacia Place, Notting Hill VIC 3168, Australia

OLYMPUS LATIN AMERICA, INC.

5301 Blue Lagoon Drive, Suite 290 Miami, FL 33126, U.S.A.

OLYMPUS KOREA CO.,LTD.

8F Olympus-Tower, 446 Bongeunsa-ro, Gangnam-Gu, Seoul, Korea 135-509