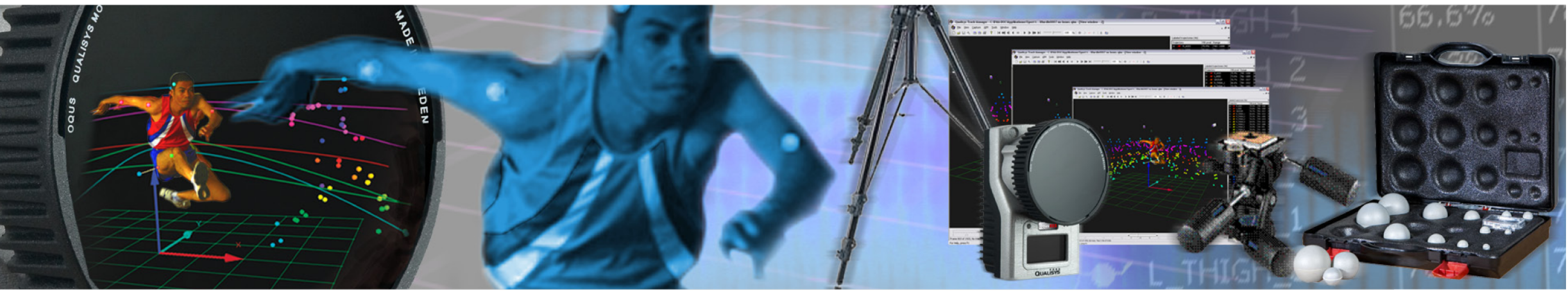
QUALISYS
Motion Capture Systems

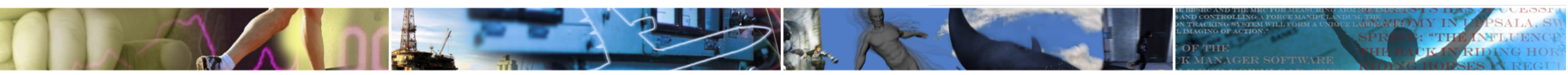


QTM

Qualisys Track Manager

USER MANUAL

QUALISYS AB
Packhusgatan 6 · 411 13 Gothenburg · SWEDEN
Tel. +46 31 336 94 00 · Fax. +46 31 336 94 20
e-mail: sales@qualisys.com · www.qualisys.com



Qualisys Track Manager, 2011-06-29

Qualisys AB

www.qualisys.com

Table of Contents

Getting Started

Qualisys Track Manager

Introduction

Qualisys Track Manager is a Windows-based data acquisition software with an interface that allows the user to perform 2D and 3D motion capture. QTM is designed to provide both advanced features required by technically advanced users and a simple method of application for the inexperienced user. Together with the Qualisys line of optical measurement hardware, QTM will streamline the coordination of all features in a sophisticated motion capture system and provide the possibility of rapid production of distinct and accurate 2D, 3D and 6D data.

During the capture, real time 2D, 3D and 6D camera information is displayed allowing instant confirmation of accurate data acquisition. The individual 2D camera data is quickly processed and converted into 3D or 6D data by advanced algorithms, which are adaptable to different movement characteristics. The data can then be exported to analysis software via several external formats.

Organization of the manual

This manual is organized into six main parts and five appendixes:

Getting Started
How to get started with the QTM software. This is only an overview without detailed descriptions and it includes a tutorial on how to perform a measurement in the chapter “Tutorial” on page 15.

QTM user interface
The QTM user interface, i.e. how to use windows, menus and dialogs.

Project options
The settings included in the **Project options** dialog.

Making a measurement
The actions needed to setup and execute a measurement.

Real time mode
Description of the real time mode.

Processing measurement data
Processing steps that can be performed on the captured data.

System hardware
The hardware included in a QTM motion capture system.

IR radiation notice

Safety notice

The Oqus and ProReflex camera uses short but quite strong infrared flashes to illuminate the markers. The flash is generated by LEDs on the front of the camera. The Oqus and ProReflex cameras are within the limits of the EN-60825-1-2001 Class 1 and the FDA CFR 1040.10 Class I classification, which means that the LED radiation is not considered to be hazardous.

However, any light of high intensity might be harmful to your eyes. Because infrared light is invisible to the human eye, you can be exposed to IR light without noticing. Therefore we recommend that you do not stare directly at the LEDs at a short distance for a prolonged time period when the camera is running.

System setup

Hardware requirements

The measurement system consists of the following parts:

1. A camera system (Oqus, ProReflex or MacReflex).
2. A calibration kit.
3. A stationary or portable measurement computer.
   a. The graphic board must have at least 64 MB built-in memory and support OpenGL 1.2 or higher. E.g. the integrated Intel graphics in laptop does sometime not work well with QTM.
   Note: For the best performance the graphic board should support OpenGL 2.1.
   Note: QTM will start a wizard with fixes for the graphic board, if a problem with the graphic board is detected. Follow the instructions in the wizard to fix the problem.
   b. It is recommended to have at least 2 GB of internal memory and it is required to have 2 GB for linearization of cameras.
   c. For Oqus there must be an Ethernet card that supports Ethernet 802.3.

Together with the measurement system the following additional equipment can be used:

- Analog interfaces, which allow capture of up to 64 channels of analog data (e.g. force plate data, EMG sensor data or other user specific analog data).
- Wireless EMG devices (Noraxon and Mega).
- Point grey cameras to capture synchronized video.
  Note: The computer used with the Point Grey cameras must be purchased for that purpose from Qualisys AB. this to ensure that it can handle the large amount of video data streamed by the cameras.
- DirectShow compatible DV cameras or standard USB Web camera, with which QTM can record video sequences for documentation.
- Sound sampling board for synchronized sound recording.

QTM is compatible with Windows 7 (64-bit and 32-bit) and XP Professional (32-bit). However the hardware might have other requirements see tables below:

Cameras		
	Win 7	**XP**
Oqus	Yes	Yes
ProReflex	Yes[1]	Yes[2]
MacReflex	No	Yes[3]

1. New Brainboxes board is needed to use the ProReflex cameras on Windows 7.
2. In XP the ProReflex cameras needs a Sealevel or Brainboxes communication board see chapter “Supported serial communication boards” on page 355. The

computer must also have a regular COM port to be able to install the communication board.

3. The MacReflex cameras were last tested with QTM 2.0.387.

Analog boards		
	Win 7	**XP**
USB-2533	Yes[1]	Yes
USB-1616FS	No[2]	Yes
PCI-DAS6402/16	Yes[1]	Yes
PCI-DAS1602/16	No[2]	Yes

1. With Instacal 6.1 or later.
2. No fully functional driver.

Other hardware		
	Win 7	**XP**
Noraxon Telemyo	Yes	Yes
Noraxon DTS	Yes	Yes
Mega ME6000	No[1]	Yes
Point Grey Grasshopper	Yes	Yes

1. No fully functional driver.

Plugin requirements

QTM can stream data in real time to a number of programs. Because the real time protocol is updated with new versions of QTM, you have to make sure that the version of the external program works with the current version of QTM. QTM supports the following programs:

Visual3D
Supplied by C-motion and available from QTM 2.0. Check with C-motion which version to use with your version of Visual3D and QTM.

MotionBuilder
Sold by Qualisys AB and available from QTM 2.1. The Motionbuilder version must be version 7.5 extension Pack 2 or later. Motionbuilder must run on Windows for the plugin to work.

Matlab and Simulink
Sold by Qualisys AB and available from QTM 2.2. The Matlab version must be Matlab 2009 or later. The Matlab program must run on Windows for the plugin to work.

LabView
Sold by Qualisys AB and available from QTM 2.2. The LabView version must be LabView 8.0 or later. The LabView program must run on Windows for the plugin to work.

iOS
Download the apps for free from Apple App Store and available from QTM 2.5. The Viewfinder app works on iOS 3.2 or later. The Remote app works on iOS 4.0 or later. The Viewfinder app can use the higher resolution of iPad and iPhone 4.

Hardware installation

The following hardware must be installed to use the QTM software:

- For Oqus, a network card with correct settings, see chapter “Network card setup” on page 341.
- For ProReflex, a serial communication board, see “Serial communication devices - Technical specification” on page C - 1.
- A camera system, see chapter “Cameras” on page 329.

There is also some optional hardware which can be used with QTM.

- An analog board for analog data from different analog units, such as force plate and EMG sensors, see “Analog capture hardware” on page E - 1.
- A wireless EMG device, see “Wireless EMG systems” on page F - 1.
- An external trigger to start the measurement, see chapter “How to use external trigger” on page 375.
- A web camera, see chapter “Video capture” on page 365.

Software installation

Make sure you are logged in with an administrator account before you start installing QTM. To install the software, insert the QTM installation CD in the CD drive. If the installation program does not auto-start, locate and execute the setup.exe file on the installation CD. Follow the instructions given during the installation.

During the installation you can select the components that you want to include. The following components can be selected: Qualisys DHCP server (required for Oqus), Noraxon EMG driver, Mega EMG driver, Point Grey camera driver and Real-time protocol documentation & example.

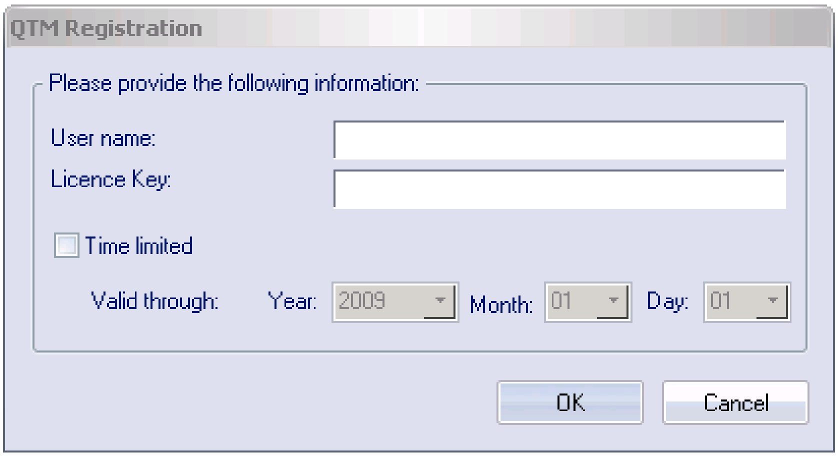
Note: QTM will also install the DHCP server called QDS. This is only used with Oqus cameras, if you are running ProReflex QDS can be disabled.

Enter the user name and the license id, which are located on the inside of the front cover of the QTM installation CD, see chapter “QTM registration” below.

If there is an internet connection, QTM will automatically check for updates when it is started. You can also use the **Check for updates** option on the **Help** menu. If the QTM computer does not have an internet connection, you can look for the latest software from another computer on the web page http://www.qualisys.com/login.html. Log in with your user name and license key. The user name and license key can also be found in QTM. Go to **About Qualisys Track Manager** on the **Help** menu.

QTM registration

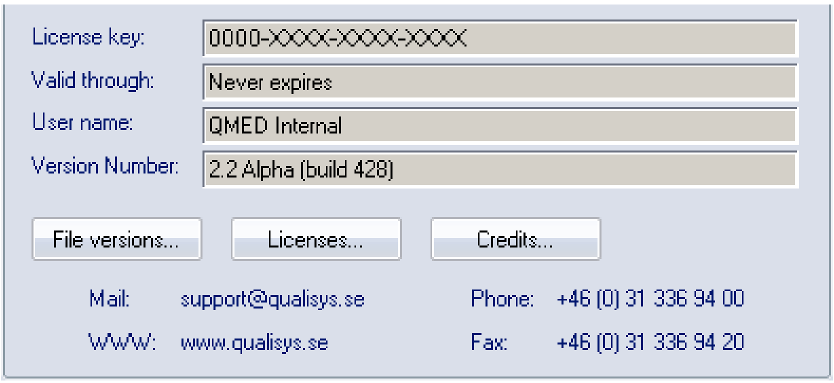
The first time QTM is started you must enter a user name and a license key. This is provided on the front cover of the QTM installation CD.



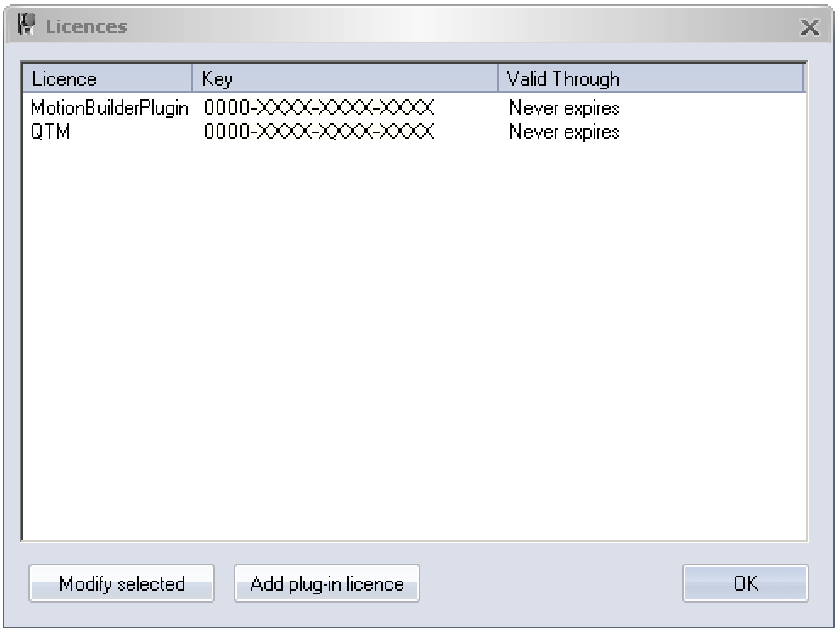
Note: If the license is time limited you must check the **Time limited** checkbox and enter the correct expiration date.

Enter plug-in license

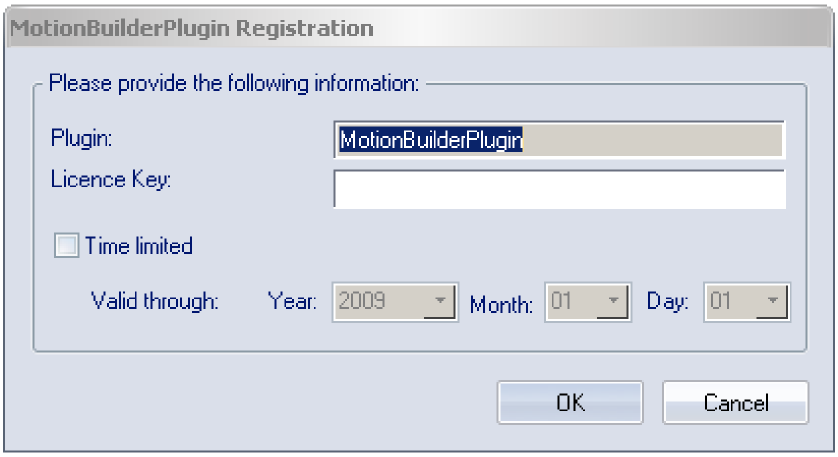
For some plug-ins the license request will appear when you start QTM. In those cases just enter the user name and license key in the dialog. However other plug-ins must be installed after QTM has started, e.g. the MotionBuilder plug-in. To enter a plug-in license in QTM click on **About Qualisys Track Manager** in the **Help** menu.



In the **About Qualisys Track Manager** dialog you can see information about the current version of QTM. Click on **Licenses** to view the installed licenses and add new licenses.



Click on **Add plug-in license** to install a new license.



Enter the license key in the dialog and then click **OK**.

Note: If the license is time limited you must check the **Time limited** checkbox and enter the correct expiration date.

Projects

QTM needs a project to capture measurement data. The project is a folder that contains all the files and information needed for QTM to process the data. A project can therefore be easily transferred to for example another computer with all the settings and files needed for the processing. To create and use projects follow the instructions in the chapters below.

Using projects

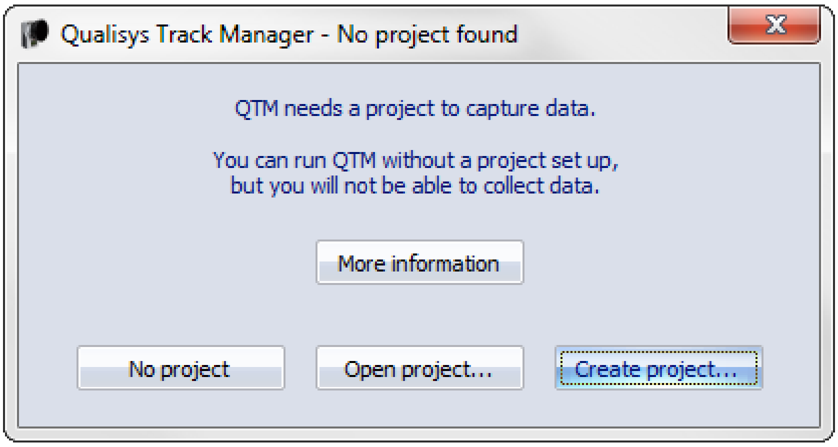
The projects can be used in a lot of different ways. Whichever way the projects are organized it is recommended to use one project per marker setup. The points below are examples of different ways to use projects.

- Use one project for all measurements with the same marker setup. Create sub-folders for the subjects under the Data folder. This also makes it easier to train the AIM model with the **Add to existing model** option.

- Create a project for each user of QTM. In this way each user can handle their own project and settings.
- The default project folder is placed in My documents. If you want several computer users to access the same project, it must be saved in a folder which all users have access to. For example on Windows 7 you can use the folders of All users.
- The project settings are saved automatically when they are changed in the **Project options**. If you want to keep a certain set of project settings you must use the backup feature, see chapter “Backup of project settings” on page 12.
- If you want to use an old workspace file (QTM 2.5 or earlier) for the project settings, you can either import it when you create the project or load it in a project with the **Import/Load old workspace file** option on the File menu.

QTM startup procedure

The first time you start QTM on a new computer you will get a question about creating a project.



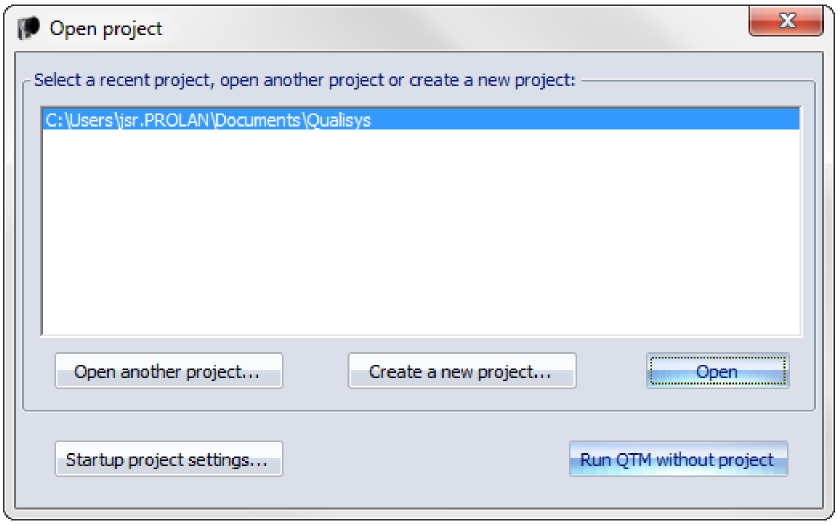
There are three ways to start QTM when you have not used projects before.

Create project
This is the default option because you must have a project to capture data in QTM.

Open project...
Use this option to open a project folder that has been copied from another computer.

No project
If you only want to open QTM files you can start QTM without a project, but you will not be able to capture any data.

After using projects the first time the default startup procedure for QTM is to display the **Open project** dialog. Each time you start QTM you have to select the project to load, to make it less likely that someone changes the projects settings by mistake. For more information about the dialog see chapter “Switch project” on page 11.

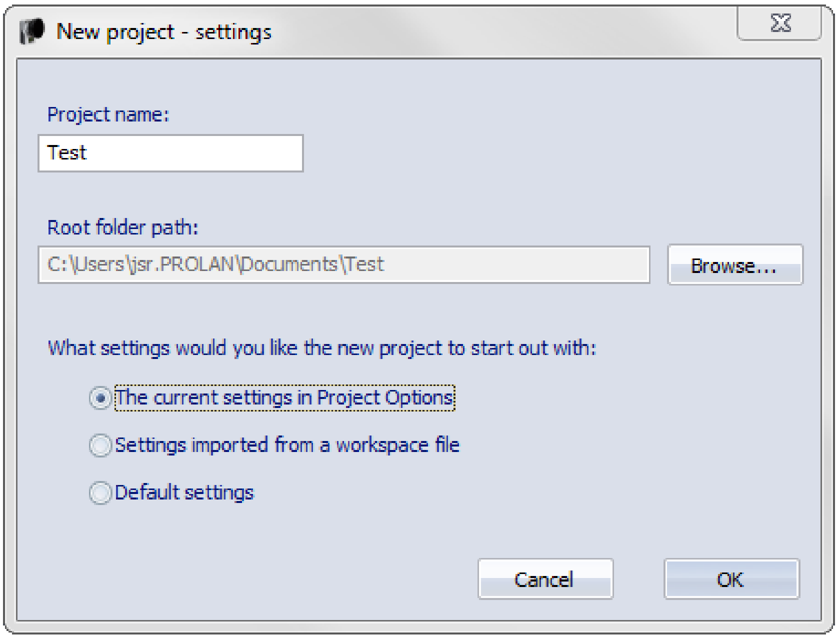


It is also possible to choose to open the most recent project or a selected project when starting QTM. To set these click **Startup project settings.**

Note: QTM will use the latest calibration made on the computer, that was made with the same cameras (placed in the same order). It means that the calibration may be loaded from another project when switching projects, it is then copied to the Calibrations folder in the project folder.

Creating a new project

A new project is created with the **New project** option on the **File** menu or in the **Switch project** dialog. The following dialog is opened with options to create a new project.



Enter the following information to create the project

Project name
The project name will also be the name of the project folder. The folder is created automatically if it does not exist at path specified in **Root folder path**.

Root folder path
This is the path of the project folder. Click **Browse** to change the path.

The current settings in Project options
The settings are copied from current settings in **Project options**. This is the default option.

Settings imported from a workspace file
The settings are imported from a workspace file saved in QTM 2.5 or earlier. Any settings that is not set in the workspace file will be set to default values.
Note: QTM will use the latest calibration made on the computer, that was made with the same cameras (placed in the same order). It means that the calibration may be loaded from another project when switching projects, it is then copied to the Calibrations folder in the project folder.

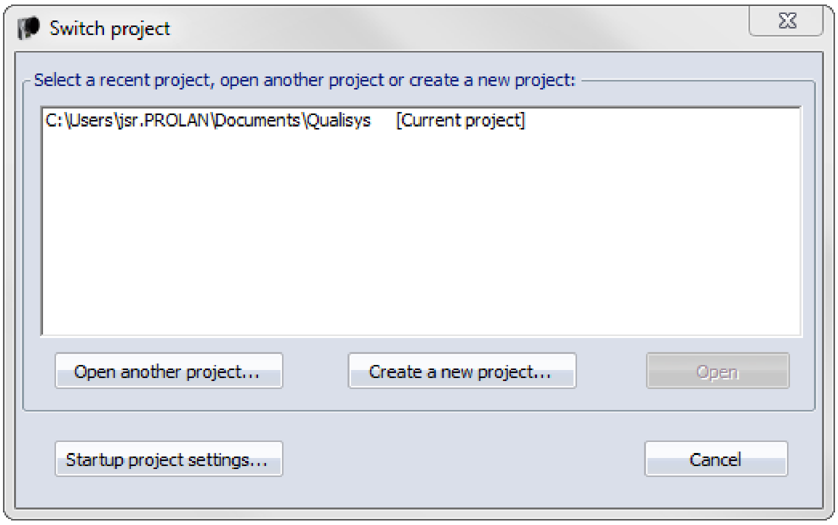
Default settings
All settings are set to the default values.

IMPORTANT: If you select any of the two last options you will lose the current settings in **Project options**. If you haven't got any projects in the recent projects list, then the settings are saved in a backup in c:\ProgramData\Workspace backups (Windows 7) or c:\Documents and Settings\All Users\Application Data\Qualisys\Workspace backups (Windows XP).

Switch project

Use **Switch project** on the **File** menu to manage the projects, for example to switch to a recently used project or open another project.
Note: Opening a project only loads the settings in **Project options**. QTM files outside the project can still be opened and processed if you like. QTM will then note that the file is not in the current project in the title bar.

Note: QTM will use the latest calibration made on the computer, that was made with the same cameras (placed in the same order). It means that the calibration may be loaded from another project when switching projects, it is then copied to the Calibrations folder in the project folder.
The calibration is also checked if the cameras are changed at **Locate** or **New**. If there is a matching calibration for the new cameras it will be loaded in the project.



The list in the **Switch project** dialog displays the 10 most recently used projects. Double-click on a project to open it. The top project is the currently open project. The current project is also displayed in the title bar of QTM and **Project options**.

The following options are available in the dialog:

Open another project
Use this option to open a project that is not in the list above, e.g. one copied from another computer.

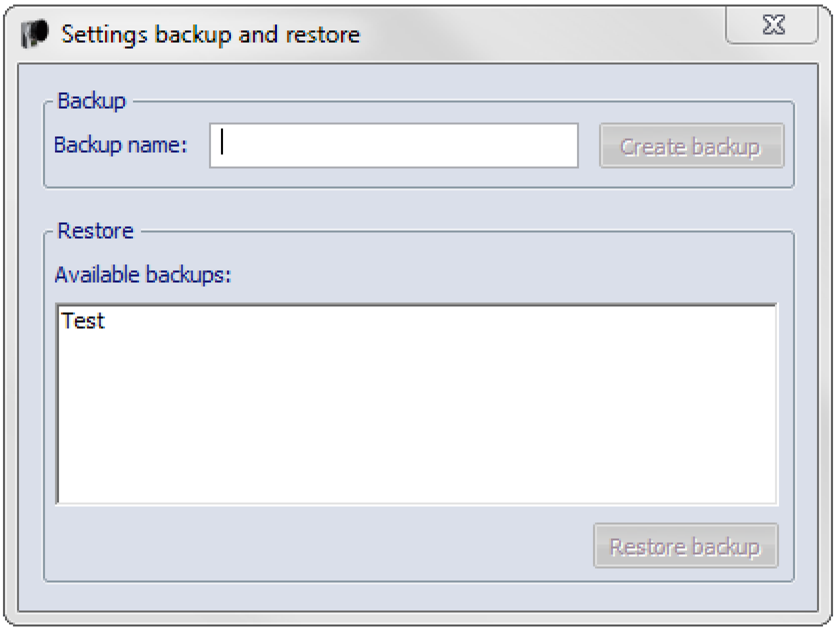
Create a new project
Create a new project, see chapter “Creating a new project” on page 10.

Open
Open the project selected in the list above.

Startup project settings
Open settings for the startup of QTM, see chapter “Startup” on page 225.

Backup of project settings

The project settings can be saved in a backup file. The backup can then be used if the settings in QTM have been changed or if you want to use two different sets of settings in the same project. Click on **Settings backup and restore** in the **File** menu to manage the backups.



Backup
Enter the **Backup name** and click **Create backup** to save a backup of the current settings in **Project options**.

Restore
The list displays all of the backups saved in the current project. Select one and click **Restore backup** to copy the settings to **Project options**.

Note: The current settings in Project options will be replaced.

Note: QTM will use the latest calibration made on the computer, that was made with the same cameras (placed in the same order). It means that the calibration may be loaded from another project when switching projects, it is then copied to the Calibrations folder in the project folder.

Project folder

The project folder is used for organizing a QTM project. It contains the following files and folders:

Data
This is the location for the captured QTM files. You can create subfolders in this folder if you want to sort the files, for example for different subjects.

AIM models
This folder contains all of the AIM models created in the current project.

Calibrations
This folder contains all of the calibrations made in the current project.

Labels
This is the default folder for Label lists.

Settings
This folder contains the backups of project settings.

Settings.qps
This file contains the current settings of the project.

QTM creates a default project folder called Qualisys in My Documents automatically when installing QTM on a new computer. However you can create a new project folder anywhere. For example if you want different users on the computer to access the same project it can be saved under the Public user in Windows 7. Copy the whole folder if you want to share the project with someone else.

There are other settings and files which are needed for all users that use QTM on a computer. To see where these are saved check the **Folder options** page in the **Project options**, see chapter “Folder options” on page 224.

Note: The QTM program and other components installed by the QTM installer are placed in Qualisys Track Manager folder under \Program Files\Qualisys (\Program Files (x86) for Windows 7 64 bit).

Windows user permissions

When using QTM it is best to be logged in with administrator rights. However on Windows 7 there are some operations that requires elevation, e.g. saving/loading workspaces. QTM also works if you are just a regular user, but there are some changes compared with being administrator.

XP
On XP the only difference compared to being administrator is that the files created by QTM (e.g. calibration files) are always stored in the 'My Documents\Qualisys' folder.

Windows 7
On Windows 7 you can be a regular user but the user must have rights to 'Back up files and directories'. This is for the elevation to work when saving/loading workspaces. To give the user this right follow the instructions:

1. Log in as administrator.
2. Open the Local Security Policy.
3. Go to Local Policies and User Rights Assignment.
4. Add the group of users to the policy 'Back up files and directories'.

Hardware configuration

After the software installation the hardware settings must be configured. Click on **Project options** on the **Tools** menu and check the following settings:

- Connection to the camera system, see chapter “Connection of the camera system” on page 227.
- Linearization of the cameras, see chapter “Linearization of the cameras” on page 229.

Check the following settings only if that specific hardware is used.

- Analog board, see chapter “How to use analog boards” on page 356.
- Force plate, see chapter “How to use force plates” on page 359.
- External trigger, see chapter “How to use external trigger” on page 375.
- Web camera, see chapter “Video capture” on page 365.

Tutorial

This tutorial goes through the fundamentals of QTM and is a summary of the chapters “Making a measurement” on page 227 and “Processing measurement data” on page 277.

The measurement in the tutorial is called ’reach and grasp’ and is an application method within psychology, which is used in the study of positions and reactions of human movement. The measurement procedure is, however, similar to any other measurement.

To perform this measurement you must have a camera system that is configured in a correct way and operational. If the camera system has not been set up, see chapter “Setting up the system (Oqus)” on page 329 or “Setting up the system (ProReflex” on page 347 for instructions on how to do so. The tutorial is written for ProReflex but most of it can be applied on Oqus.

The following equipment is needed to perform the task:

1 Qualisys 3-camera system or larger

1 Measurement computer with QTM installed

1 Table

1 Chair

1 Cup (the measurement object)

8 Reflective markers (preferably 12 mm)

1 Calibration kit (wand 300 or 750)

1 Test subject

Task description

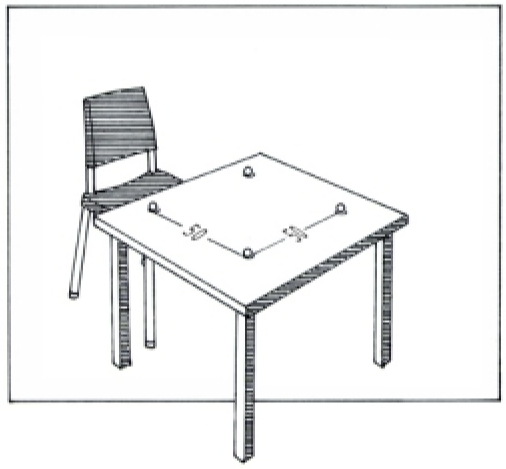
When the measurement starts the test subject will reach for the cup, move it from position A and place it at position B.

Measurement and processing – Step by step

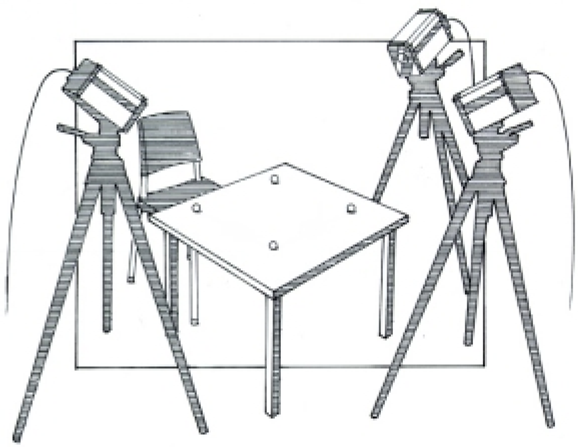
Below follows a step by step list of how to perform a measurement.

Measurement setup

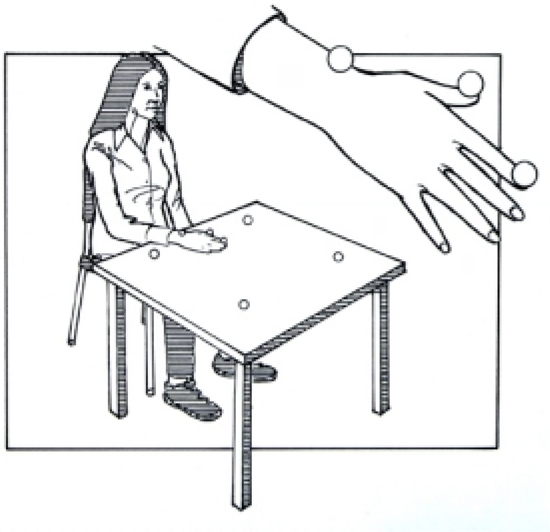
1. First the measurement volume is specified. Place a square of four markers (50 x 50 cm) on the table. The markers represent the outer corners of the base of the measurement volume.



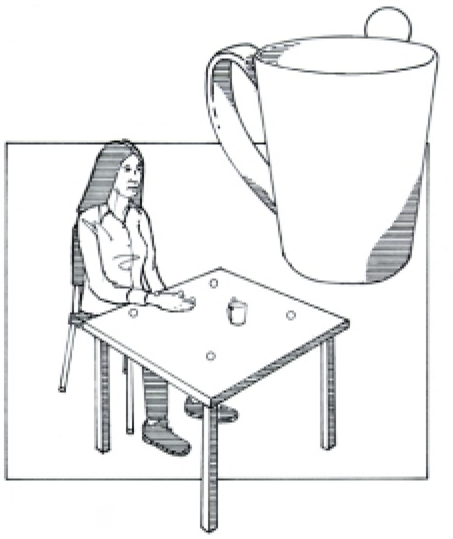
2. Mark the two positions called A and B with tape, they can be placed anywhere in the measurement volume. The positions are needed to make the repetition of the measurement easier.
3. Place the three cameras around the measurement volume, according to the picture below. Turn on the camera system on the master MCU, i.e. the camera where the yellow data cable is connected.



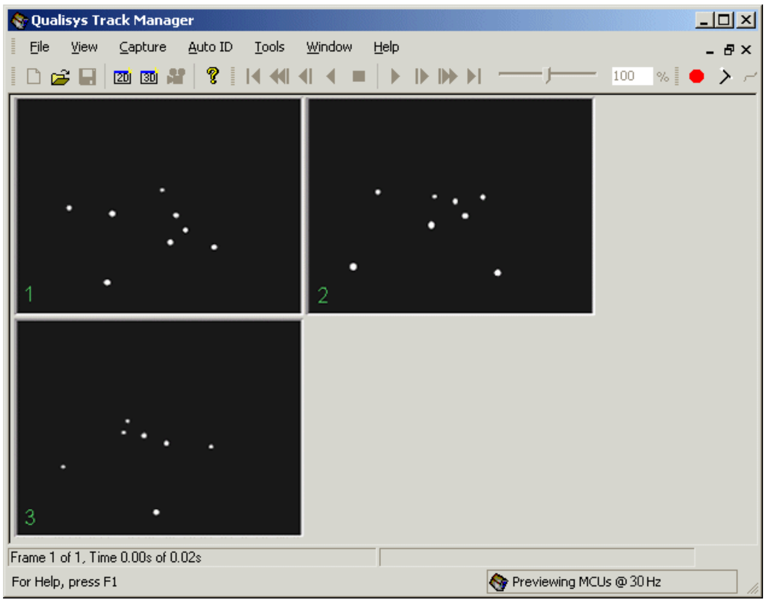
4. Apply the markers on the test subject, according to the picture below, place the test subject on a chair in front of the measurement volume.



5. Apply the marker on the measurement object, according to the picture below, and put it on position A.



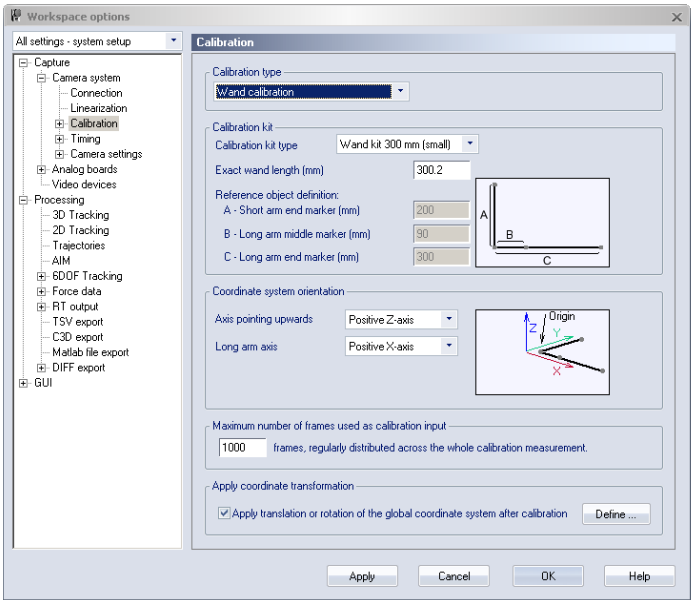
6. Start QTM from the **Windows start** menu and open a new capture file in QTM, so that you can see a 2D view, in the preview mode, from each camera.
   Note: If the camera system is not found by QTM, it must be located on the **Connection** page in the **Project options** dialog, see chapter "Camera system connection" on page 227.



7. Adjust the cameras, one at time, so that every camera can see all of the markers, both on the test subject and on the measurement object.
8. Open the **Project options** dialog and set the frame rate to 120 Hz on the **Camera system** page.
9. Adjust the focus and aperture, according to the description in “Aperture and focus” on page 349.
10. Ask the test subject to perform the task described above. Make a quick check that all markers are visible during the test and that the intensity is OK, i.e. that the markers’ **xSize** and **ySize** for each camera in the Data info window are at least 5 to 7.
    Note: For Oqus this is above 200 subpixels.
11. Ask the test subject to step out of sight of the cameras. Remove the markers that represent the measurement volume.
    Tip! Mark the corners with tape.

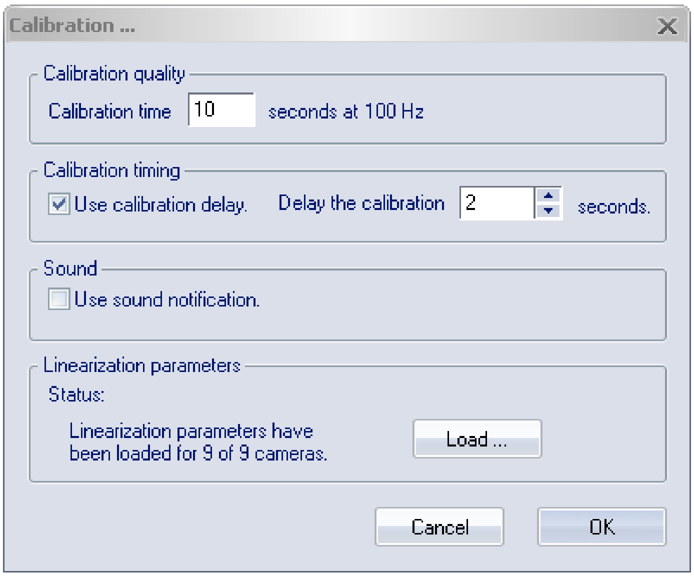
Calibration of the system

1. Place the calibration structure in the measurement volume, so that all of the cameras can see the four markers on the calibration structure.
2. Check the **Calibration** settings in the **Project options** dialog. I.e. click **Project options** on the **Tools** menu and then click **Calibration**. Make sure that all of the settings are set according to the following picture. Click **OK** to return to the preview window.
   Note: The exact wand length is individual to each calibration structure. Check the plate on the wand for the **Exact wand length** and enter the correct value.

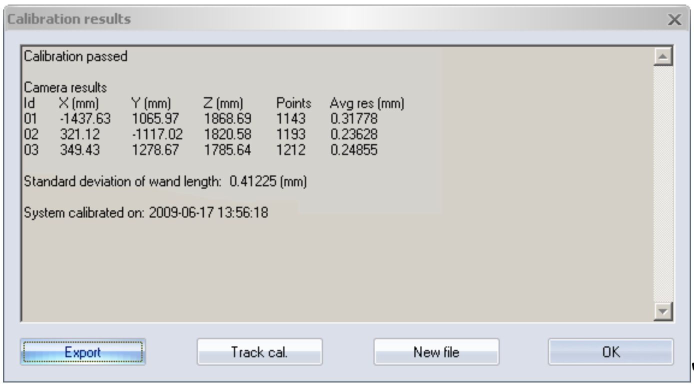


Note: If you have used wand 750 instead, the setting for **Calibration kit type** must be changed.

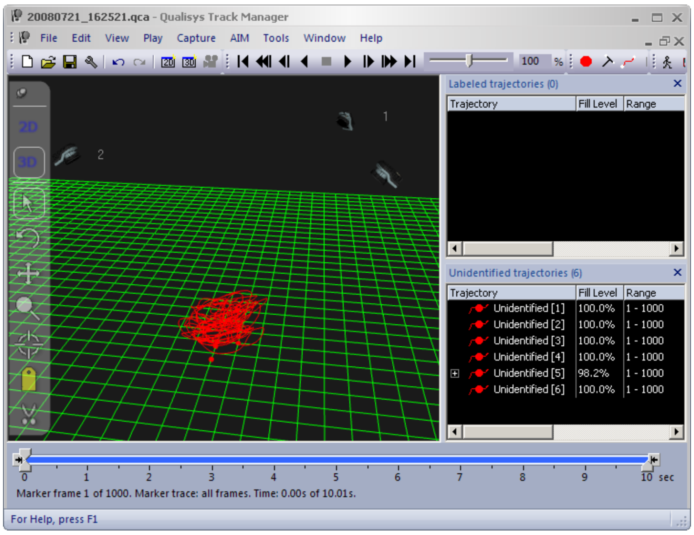
3. Click the **Calibration** icon or click **Calibrate** on the **Capture** menu to open the **Calibration** dialog.
4. Check that the linearization files of the cameras are installed, i.e. look under the **Linearization parameters** heading. If the files are not installed click on **Load** and follow the instructions in the chapter “Linearization of the cameras” on page 229.
5. Enter the calibration time, 10 s, under the **Calibration quality** heading. Click **OK** to start the calibration capture. For a description of how to perform the calibration see chapter “Wand calibration method” on page 241.



6. When the calibration is done a window with the calibration results is shown. The results will tell you if the calibration passed and some calibration quality results.



7. You can also see how well you have covered the measurement volume by clicking **Track cal**.



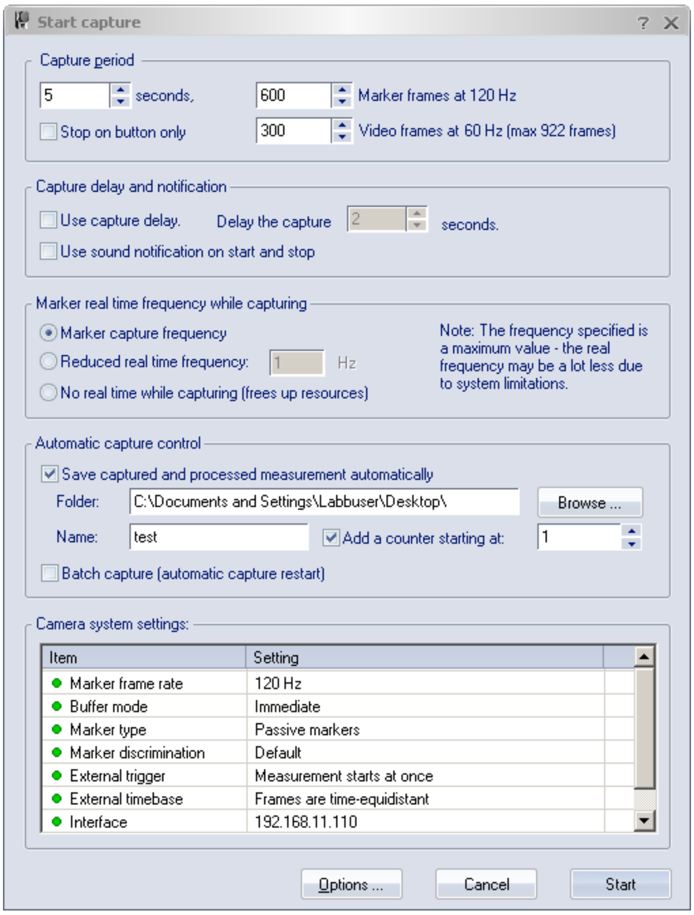
8. Close the calibration file by clicking **Close** on the **File** menu. Remove the calibration object. The measurement volume has been calibrated and the measurement can start. Ask the test subject to be seated.

Start a new capture

1. Start a new capture file in QTM and click the **Capture** icon or click **Capture** on the **Capture** menu. Set the **Capture period** to 6 seconds and click **Start**. The capture will start immediately if no capture delay is specified. When clicking **Start** tell the test person to perform the task.

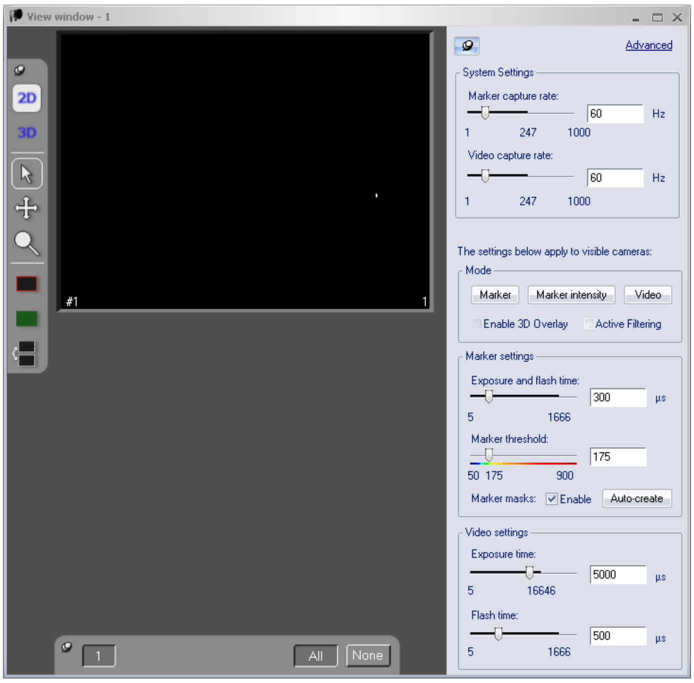
   Note: If the check box under the **Previewing while capturing** heading is selected the measurement can be viewed in a **View** window during the capture.

   Note: To automatically track the 3D data, **Track the measurement** and **3D** must be selected on the **Processing** page in the **Project options** dialog.



2D and 3D view windows

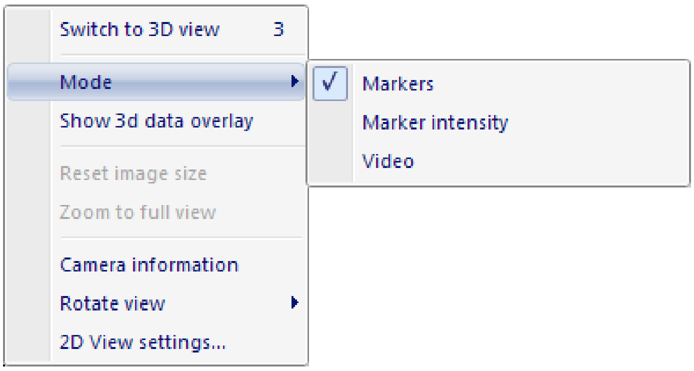
1. When the capture is done you can view the result in both 2D and 3D views.



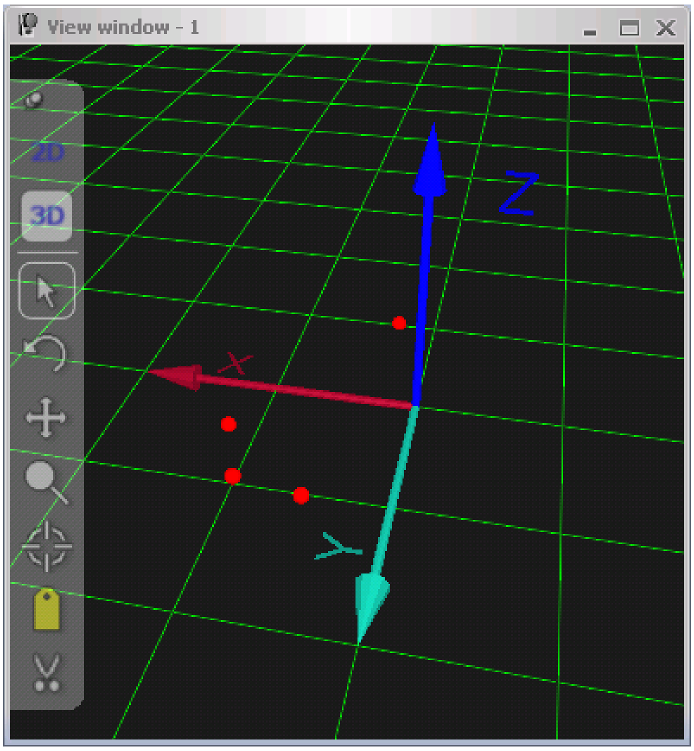
2. In the 2D view you can see the captured view from each camera during the measurement. To play the capture file click the **Play** icon on the **Playback** toolbar.

   Right-click in the 2D view to open the **View window** menu, which contains options for the view. Some of the options on this menu are not available from the 2D view and they will therefore be gray. For more information about the menu see chapter “2D view window” on page 36.

3. Switch to the 3D view on the **View window** menu or open a new 3D view window by clicking the **3D view** icon .

4. In the 3D view the measurement is shown in 3D. You can see the whole measurement by playing the file in the same way as in the 2D view. The measurement can be seen from different perspectives by using the zoom, rotation and translate functions. The functions are used by moving the mouse while holding down the following mouse button combinations:

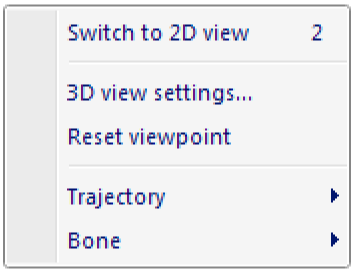
   Left-click – Rotation

Right-click – Translation

Right- and left-click at the same time or middle-click – Zoom

For more information about how to use the **3D view window** and how to change the graphics see chapter “3D view window” on page 50.

In the 3D view there are more options on the **View window** menu than in the 2D view, for information about the options see chapter “3D view window menu” on page 60.



Trajectory identification

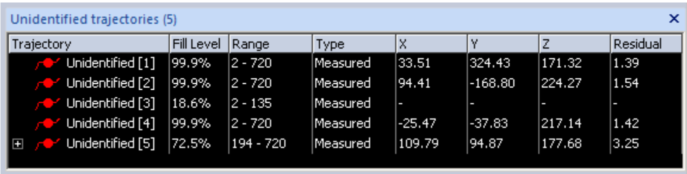
1. To the right of the **View** windows there are three **Trajectory info** windows:

   Labeled trajectories window

   Unidentified trajectories window

   Discarded trajectories window

   After a measurement all markers’ trajectories will appear in the **Unidentified trajectories** window.



Unidentified trajectories (5)

Trajectory	Fill Level	Range	Type	X	Y	Z	Residual
Unidentified [1]	99.9%	2 - 720	Measured	33.51	324.43	171.32	1.39
Unidentified [2]	99.9%	2 - 720	Measured	94.41	-168.80	224.27	1.54
Unidentified [3]	18.6%	2 - 135	Measured	-	-	-	-
Unidentified [4]	99.9%	2 - 720	Measured	-25.47	-37.83	217.14	1.42
Unidentified [5]	72.5%	194 - 720	Measured	109.79	94.87	177.68	3.25

The columns of the trajectories contain the following data:

Trajectory
The label of the trajectory and the color of its marker in the 3D view.

Fill level
The percentage of the measurement range that the trajectory or part is visible. The measurement range is selected on the **Timeline control** bar.

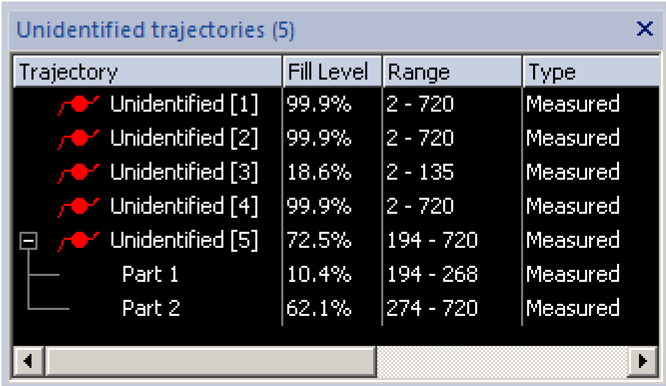
Range
The range of frames, within the measurement range, with data for the trajectory or part.

Type
The type of trajectory (**Measured**, **Mixed** or **Gap-filled)**.

X, Y and Z
The position (in mm) of the trajectory in the current frame. The distance is to the origin of the coordinate system of the motion capture.

Residual
The average of the residuals of the 3D point. This is a quality check of the point’s measured position.

Because of the visibility problems some trajectories may consist of more than one part. If this happens a plus sign is visible in front of the trajectory. The reason for these parts is that a marker has been obscured during a measurement, which results in two trajectories. **Auto join** tracking then joins the two trajectories automatically into one trajectory with two parts, click on the plus sign to view the parts.



Unidentified trajectories (5)

Trajectory	Fill Level	Range	Type
Unidentified [1]	99.9%	2 - 720	Measured
Unidentified [2]	99.9%	2 - 720	Measured
Unidentified [3]	18.6%	2 - 135	Measured
Unidentified [4]	99.9%	2 - 720	Measured
Unidentified [5]	72.5%	194 - 720	Measured
Part 1	10.4%	194 - 268	Measured
Part 2	62.1%	274 - 720	Measured

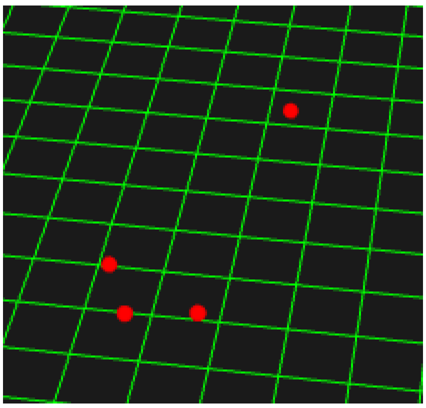
However, sometimes the **Auto join** tracker fails and an extra trajectory for that marker is created. That is the reason why there are more trajectories than markers in this example. In an ideal capture there would be the same number of unidentified trajectories as the number of markers that are used in the setup.

2. It is time to identify the unidentified trajectories. There are two ways to identify trajectories in QTM: manual or automatic identification. They are both described below, first manual identification and then the automatic in step 6.

3. There are several ways of doing a manual identification. In this section just one of them is described.

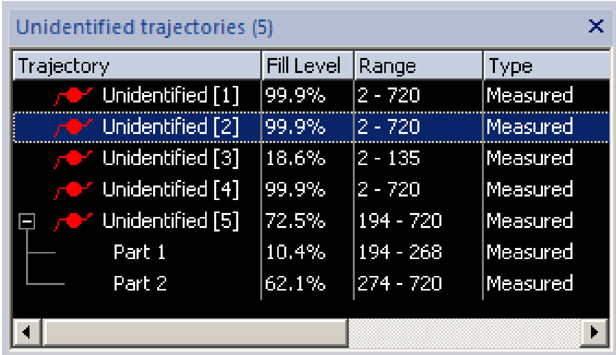
    a. Go to a frame in the capture file where all of the trajectories (markers) are visible.

    Tip! It can be easier to do the identification if the text labels and axes are not visible in the 3D view.

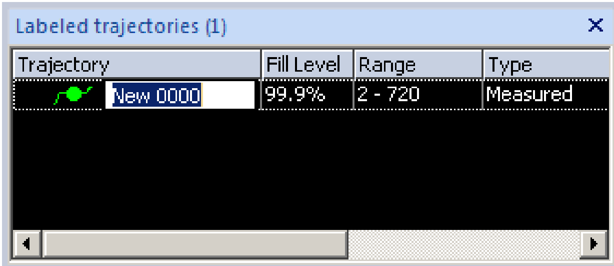
    

    b. Select the trajectory that represents the cup by clicking its marker in the 3D view. The marker will be white when it is selected.

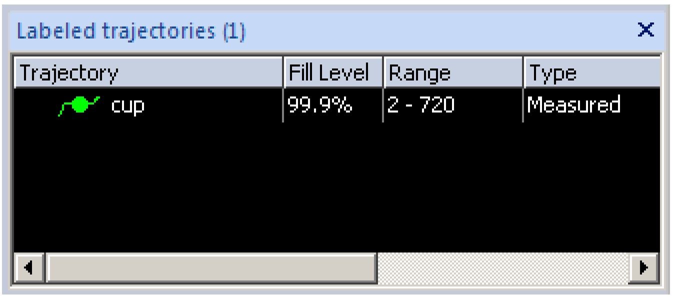
    At the same time the trajectory will also be selected in the **Unidentified trajectories** window.



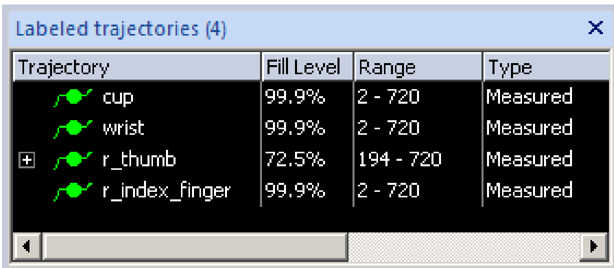
c. Drag and drop the selected trajectory by holding down the left mouse button and move it up to the **Labeled trajectories** window. The moved trajectory will be green and labeled 'New 0000'.



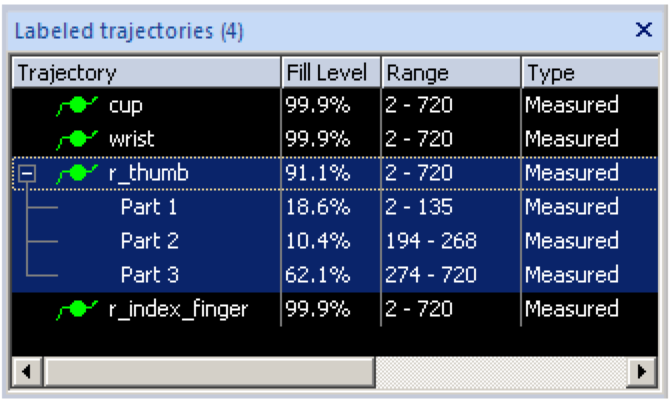
d. Double-click the text label 'New 0000' and enter the correct name 'cup' for this trajectory.



e. Repeat the steps b, c and d until all unidentified trajectories are labeled. Use the same labels as in the picture below.

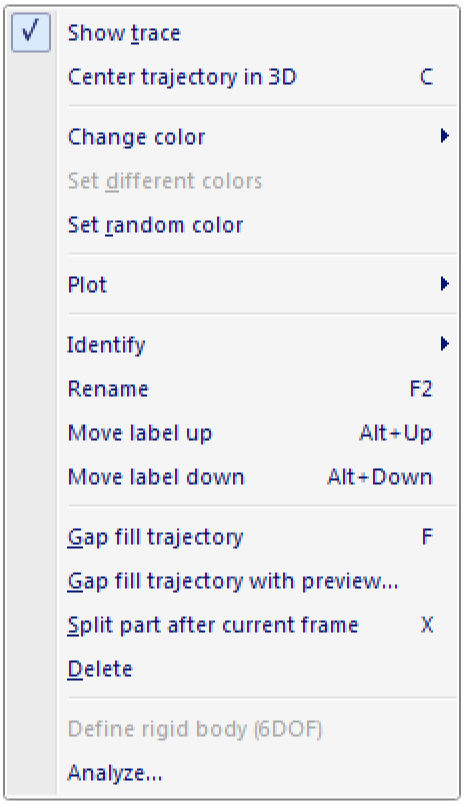


f. If there are more trajectories than markers you need to combine these trajectories manually with each other. This is done by a simple drag and drop of the extra trajectory to the trajectory with the correct name in the **Labeled trajectories** window. Afterward the trajectory will consist of more than one part.



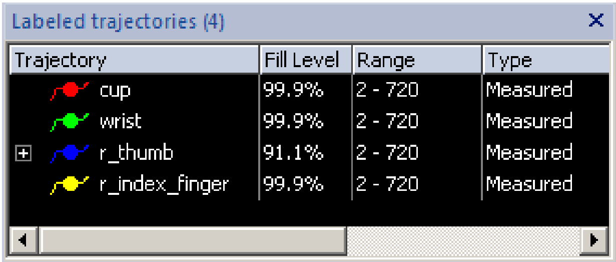
Note: Another way to perform a manual identification is to first load a label list in the **Labeled trajectories** window by clicking **Load label list** in the **Labels** menu. Then drag and drop the unidentified trajectories to the correct label.

4. Right-click on a trajectory in the **Trajectory info** window to open the **Trajectory info window** menu. The options are applied to the trajectories that are selected. For information about the menu see chapter “Trajectory info window menu” on page 69.

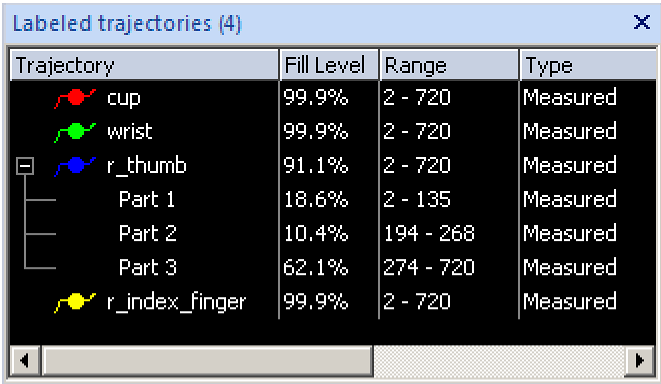


5. This step is a description of how to use the **Gap fill trajectory** function. The example is done by gap filling the trajectory parts of the 'r_thumb' trajectory. In your measurement it might be another trajectory which can be gap filled. For more information about gap fill see chapter “Gap fill trajectories” on page 72.

   a. It is easier to separate the different trajectories if they have different colors. Therefore it is a good idea to change the colors with the option **Set random color**. Make sure that all trajectories are selected before clicking **Set random color**, use Shift + Left-click to select the trajectories. Set the colors manually if the random colors are too similar.

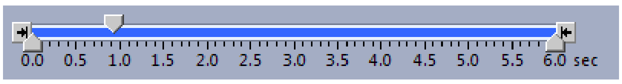


   b. Expand the 'r_thumb' trajectory by clicking on the plus sign, so that all of the parts in the trajectory are shown.

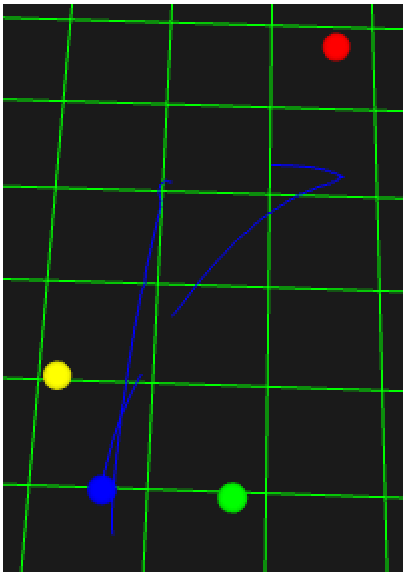


   c. Toggle off the trace of the other trajectories in the 3D view, by selecting them and then click **Display trace** on the **Trajectory info window** menu. This is just to make it easier to see the result. It can also help to disable the display functions for axes and text labels.

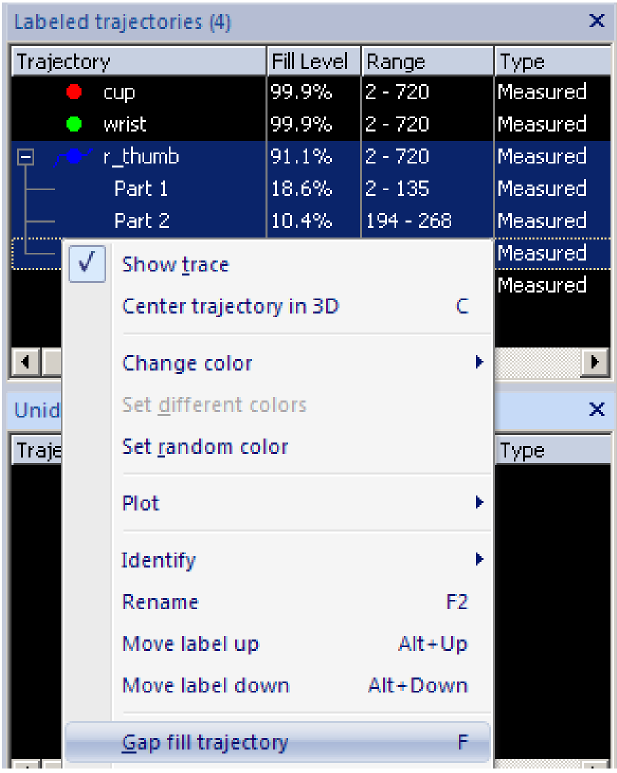
   d. Make sure that the trace is visible in the 3D view and that the trace range is expanded to the maximum (i.e. check that the bottom sliders on the **Timeline control** bar are as far apart as possible, see picture below).



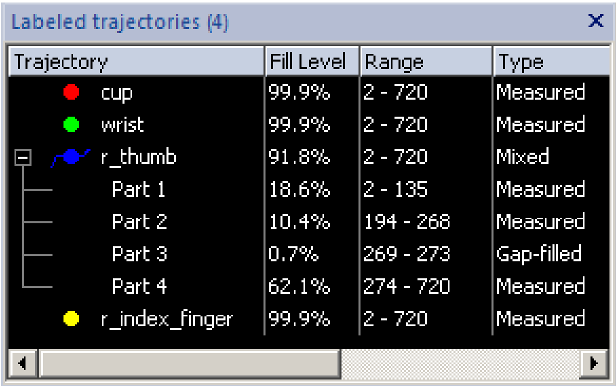
   e. As you can see it is quite easy to see the gap between the different parts of the 'r_thumb' trajectory.



f. To gap fill the trajectory, right-click on 'r_thumb' to open the **Trajectory info window** menu. Click **Gap fill trajectory**.



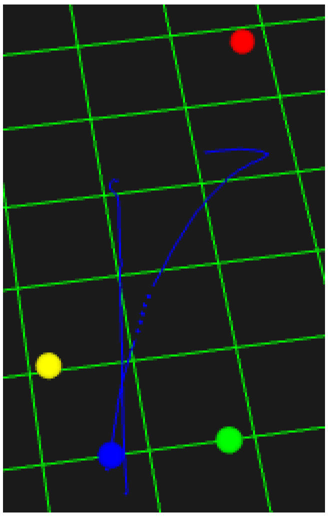
g. A fourth part is created, which is the output of the **Gap fill trajectory** function.



Labeled trajectories (4)

Trajectory	Fill Level	Range	Type
cup	99.9%	2 - 720	Measured
wrist	99.9%	2 - 720	Measured
r_thumb	91.8%	2 - 720	Mixed
Part 1	18.6%	2 - 135	Measured
Part 2	10.4%	194 - 268	Measured
Part 3	0.7%	269 - 273	Gap-filled
Part 4	62.1%	274 - 720	Measured
r_index_finger	99.9%	2 - 720	Measured

h. The gap filled part is shown as a dotted trace in the 3D view.



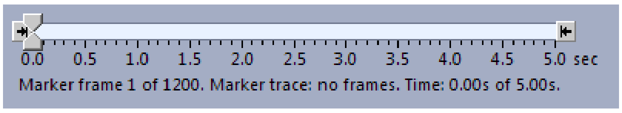
6. To use automatic identification you must first generate an **AIM** model, which can then be applied on unidentified measurements with the same marker setup and similar motions. For more information about automatic identification see chapter "Automatic Identification of Markers (AIM)" on page 302.

   a. First you need to identify the trajectories in a file according to the description above.

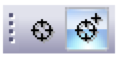
      Tip! Use the file that you have just identified.

   b. Make sure that the whole measurement range is included in the model, i.e. check that the scroll boxes on the **Timeline control** bar are as far apart as possible. This is necessary to include the whole motion and make the model as accurate as possible.

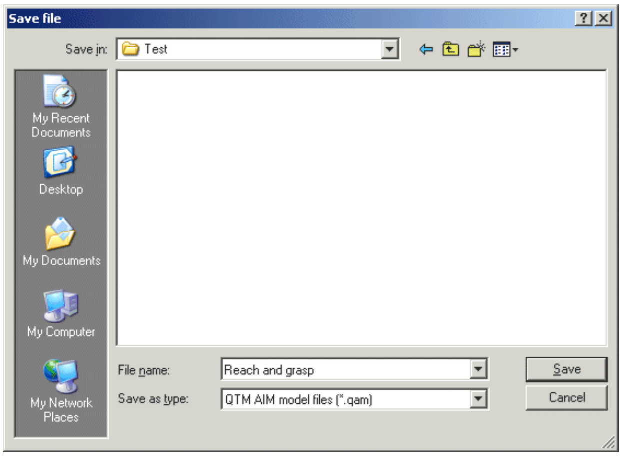
      Note: If the data in the file is erratic or some trajectories have large gaps that cannot be gap-filled, it is sometimes better to just use a part of the measurement, see chapter "Generating an AIM model" on page 302.



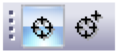
   c. Click the **Generate model** icon. The built-in auto identification function in QTM will generate an **AIM** model.



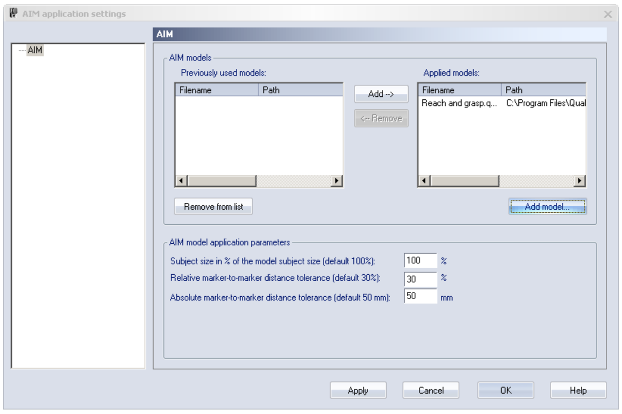
d. Select the directory where you want to save the model, e.g. the same folder as the capture files. Then enter a **File name** for the model and click **Save**.



7. You can then use this **AIM** model on unidentified measurements. Close all files and perform a new measurement with the same setup. When the measurement is done, click the **Apply model** icon.



a. A dialog is displayed with the settings on the **AIM** page in the **Project options** dialog. Make sure that the correct **AIM** model is loaded and click **OK**. The other settings do not need to be changed unless the application of the AIM model fails.



b. The file is automatically identified. To speed up the identification process even more, you can apply the **AIM** model either directly after a capture or during batch processing, see chapter “Batch processing” on page 279.

Data export to TSV

1. When all of the trajectories are labeled (and gap filled if that is necessary) in a correct way you can export the data for analysis in external software. QTM supports the following export formats:

   TSV

   C3D

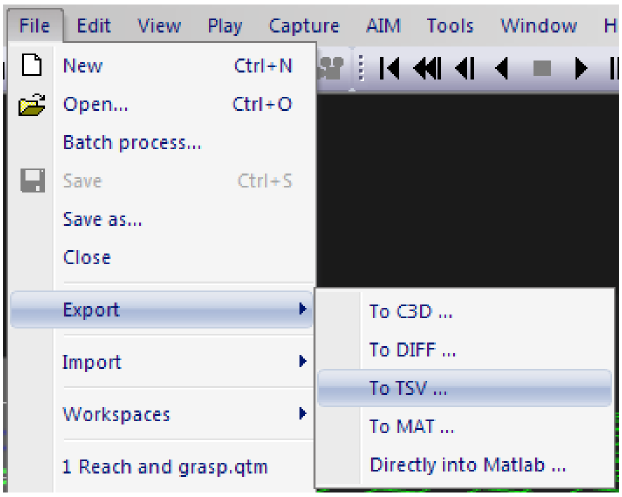
   MAT

Direct export to Matlab

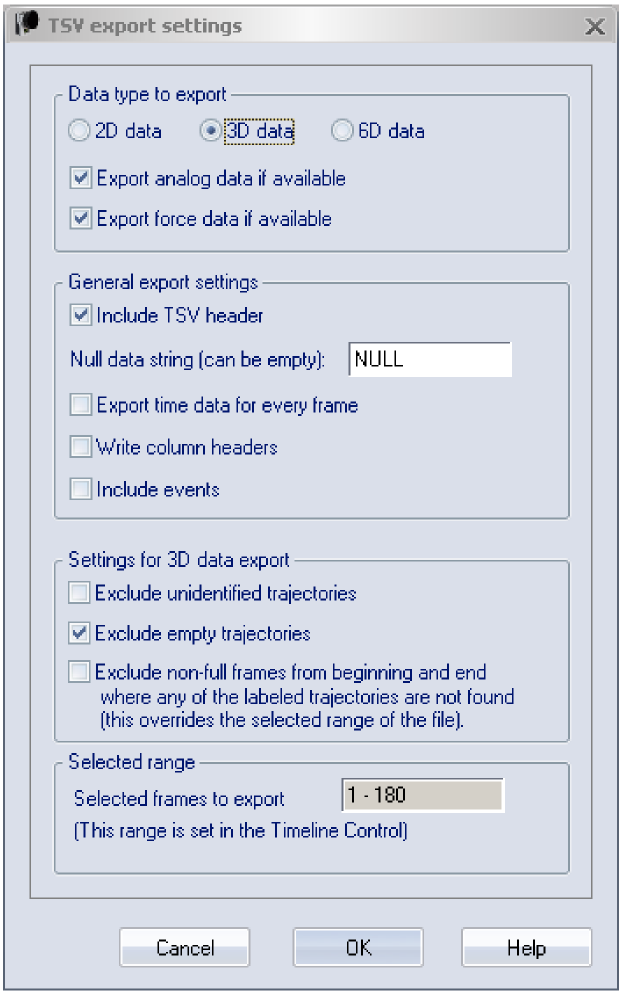
DIFF

Only export to TSV format is shown in this example. For more information about the export formats see chapter “Data export” on page 314.

a. Go to the **File** menu and click **Export** and then **To TSV**. The TSV file will only include data in the measurement range, which is defined with the boxes on the **Timeline control** bar.



b. Set the settings for the TSV export and click **OK**. For information about the settings see chapter “TSV export” on page 208.



c. The TSV file can then be opened and viewed in e.g. Excel.

QTM user interface

QTM specific user interface

RT(Preview)/File mode

The QTM software has two modes of operation: RT (preview) mode and file mode.

The RT (preview) mode is entered when a new file is opened in QTM. Click on **New** on the **File** menu to open a new file. In RT (preview) mode the motion capture is shown in real-time before and during a measurement. In RT, the data can also be accessed via a real time TCP/IP protocol.

The file mode is used when the motion capture is finished. Manual editing of a QTM file can only be done in file mode.

Window types in QTM

There are two window types in QTM: **View** windows and **Tool** windows.

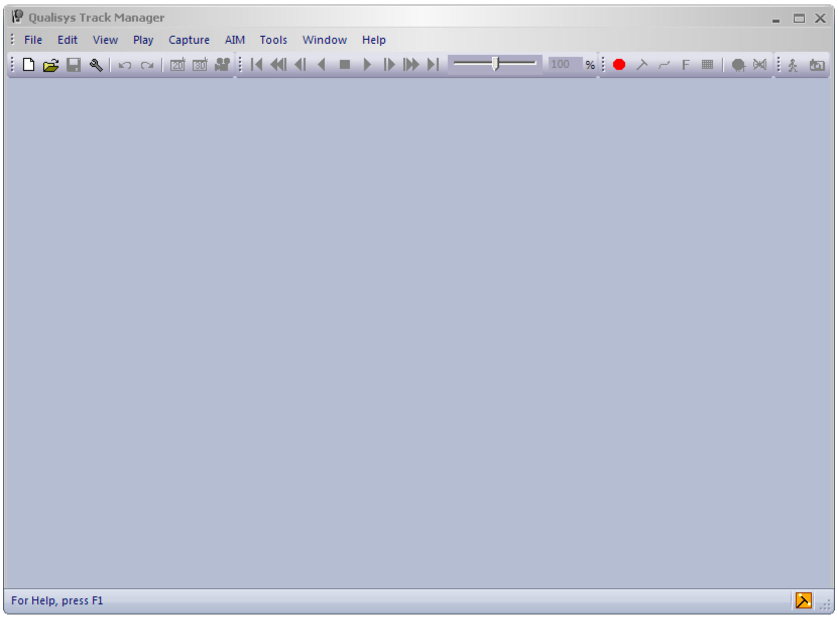
The **View** windows can be moved and resized in any way, just as a regular window in a Windows program. For example it can be tiled with commands in the **Window** menu and it can also be maximized so that it fills the main window.

The **Tool** windows can either be docked at the side of the main window or floating above the **View** windows. Double-click on the **Title** bar of the window to toggle between docked and floating.

QTM windows

Main window

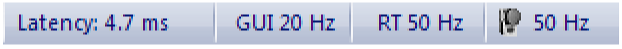
The main window is opened when QTM is started. It contains the different menus and toolbars. At the bottom of the window there is a **Status** bar, which shows important messages and progress information.



Main status bar

The main **Status** bar contains messages about what is going on in QTM, e.g. when QTM is capturing or processing data. There can also be status messages for the real time processing and the camera synchronization.

It also shows the latency and the different frequencies during real-time and when capturing a measurement. The frequencies are updated continuously so that if the computer cannot process the data fast enough the frequencies will decrease. Next to the frequencies is a symbol that shows the status of the calibration, see chapter “Introduction to calibration” on page 235.



Latency
The current real-time latency. Only displayed if enabled on the **GUI** page in the **Project options** dialog. For more information see chapter “Real time latency” on page 275.

GUI
This is the update frequency of the QTM GUI. It can be changed on the **GUI** page in the **Project options** dialog.

RT
This is how fast the data is processed by QTM in real-time. The frequency is set on the **Camera system** page in the **Project options** dialog or for a measurement in the **Start capture** dialog.
The **RT** frequency can be lower than the camera frequency in two cases. First in RT/preview and Capture mode if the camera frequency is too high so that the computer cannot process all of the data. The second case is during a measurement in capture mode if **Reduced real time frequency** is selected.

This is how fast the data is captured by the cameras. In RT/preview mode the frequency depends on the **Real time frequency** setting on the **Camera system** page in the **Project options** dialog. When **Reduced real time frequency** is selected the frequency will be displayed as **reduced in RT** in the status bar.
During a measurement the displayed frequency is always the same **Marker capture frequency**.
Note: When external timebase is used **EXT** will be displayed in front of the frequency.

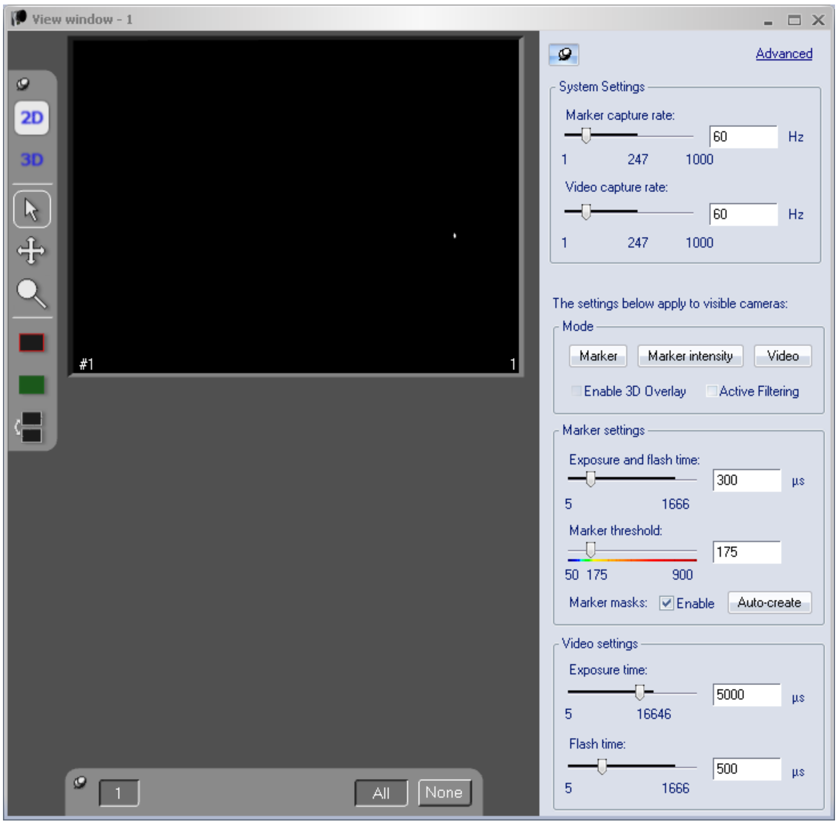
View windows

In a Viewwindow the motion capture data can be viewed in 2D, 3D view. The video data of Oqus and DV/webcam is displayed in the 2D view window.

For each view there is a **View window** menu with settings. The menu is accessed by right-clicking in a View window. There can be 30 View windows opened for a qtm-file, this also includes the **Plot** windows.

The **Timeline control** bar is common for all View windows and placed at the bottom of the QTM window, see chapters "Timeline control bar" on page 61.

2D view window



The 2D viewwindow display the raw data of all cameras connected to QTM. This includes 2D views of the motion capture data from each camera in the measurement system. It also includes Oqus video that are displayed in the 2D view of the Oqus camera and the DV/webcam video which is displayed after the motion capture cameras. The 2D view window also includes the following controls:

2D view toolbar
Contains tools for the 2D view, see chapter "2D view toolbar" on page 38.

Camera selection bar
Select the cameras that you want to view in the 2D view window, see chapter "Modifying the 2D view" on page 39.

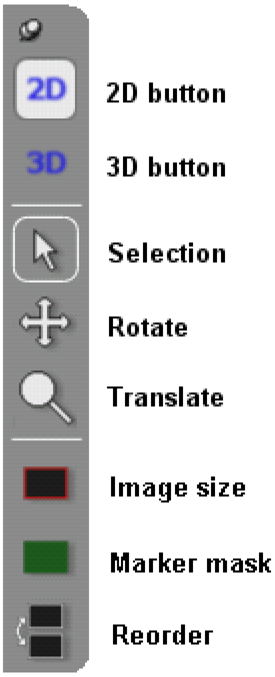
Camera settings sidebar (Oqus)
Control the basic settings for Oqus cameras, see chapter "Camera settings sidebar" on page 40.

2D view window menu
Right-click to open options for a single camera, see chapter “2D view window menu” on page 44.

The following objects are displayed in a 2D view for a camera:

- The number in the lower left corner of the 2D view of a camera is the camera id. A motion capture camera is displayed as for example #1, and a DV/webcam camera is displayed as for example #1V.
- The number in the lower right corner is the current number of markers seen by the camera. A 'V' is displayed instead, if a motion capture camera is in video mode.
- The current **Image size** of the camera is shown as a red square and the part of the image that is outside the image size is greyed out. For Oqus cameras the **Image size** can be changed with the **Image size** tool on the **2D view** toolbar, see chapter “2D view toolbar” on next page.
- For Oqus the marker masks are shown as green squares, see chapter “How to use marker masking” on page 264.
- The Oqus video is also shown in the 2D view window, both in preview and in a file. This means that all of the actions, like zoom and 3D overlay, can be performed both in preview and in a file. How to capture video from Oqus is described in the chapter “Oqus video capture” on page 256.
- The DV/webcam video are displayed after the motion capture cameras in the 2D view window. You can use zoom on the video in both preview and file mode. The video cameras will appear in the same order as they are on the **Video devices** page in the **Project options** dialog. For more information about DV/webcam video see chapter “DV/webcam video in 2D view” on page 42.
- The 2D markers are color coded when marker filtering is activated on the Oqus camera, see chapter “How to use circularity filtering and correction of merged markers” on page 267.
- There is a red warning sign in the top left corner if the camera has problems calculating the markers. Hover the mouse over the warning sign to get more information.
  The most common problem is that the camera does not have time to calculate all of the markers. This is actually caused by too much background light. If this happens, reduce the **Exposure time** or increase the **Marker threshold** on an Oqus camera or close the aperture on the ProReflex camera.
- If the markers are grey and it says **Not used for tracking** in the middle of the view, that camera has been deactivated on the **Linearization** page in the **Project options** dialog.
- The delayed exposure setting is displayed next to the camera number, e.g. **expgrp: 1** when the camera is in exposure group 1.

2D view toolbar



The **2D view** toolbar contains settings for manipulating the 2D view of the different cameras and for switching between 2D and 3D view. From left to right the icons have the following use.

2D button
Switch to 2D.

3D button
Switch to 3D.

Selection cursor
Use the normal mouse behavior.

Translate cursor
Use the left mouse button to translate the 2D view.

Zoom cursor
Use the left mouse button to zoom the 2D view.

Image size cursor
Use the left mouse button to draw a new image size for a camera. The new image size will be marked with a red frame and everything that is outside the frame is greyed out. To modify an existing image size frame use the normal cursor and drag the edges or move the whole frame. For more information about image size see chapter "Image size" on page 145.
Note: When changing the capture rates from the **Camera settings** sidebar the image size is reduced automatically if the frequency is higher than max frequency at full image size. However when changing the capture rate from the **Project options** dialog the image size must be reduced first. For example if you have one camera in video mode you still have to reduce the image size in video mode for all of the cameras.

Marker mask cursor
Use the left mouse button to draw a new marker mask for a camera, it can be done either in **Marker** or **Marker intensity** mode. The new marker mask will be added as a green area and the markers inside the mask will be removed. There can be a

maximum of 5 marker masks per camera. To modify an existing marker mask use the normal cursor and drag the edges or move the whole mask. Right-click on mask and select **Delete this mask** to delete it. For more information about marker masks, see chapter “Oqus marker masking” on page 264.

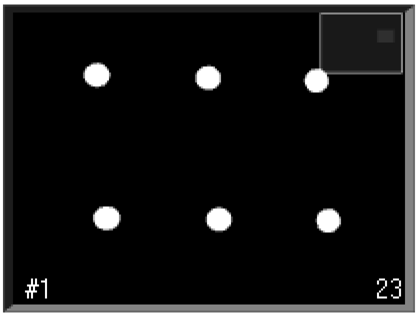
Change camera order cursor
Use the left mouse button and drag and drop the whole camera view to change the camera order in QTM. Use this cursor to change the number on the camera display so that they come in the order that you want.

Modifying the 2D view

The appearance of the 2D view can be modified with the following options.

- Camera buttons at the bottom of the window to select which cameras to display. When a button is pressed down the corresponding camera is displayed in the 2D view. To hide a camera release the button. The **All** button will show all cameras and **None** will hide all cameras.
  Note: The DV/webcam cameras are called for example '1V' and are always placed last in the list.
- Double-click in the area of a camera to just display that camera in the 2D view window. Use the arrow buttons to step to the next and previous camera in the system.
- Use the mouse or the buttons on the **2D view** toolbar to change the zoom and translation of a camera view. The 2D views can be zoomed and translated individually.

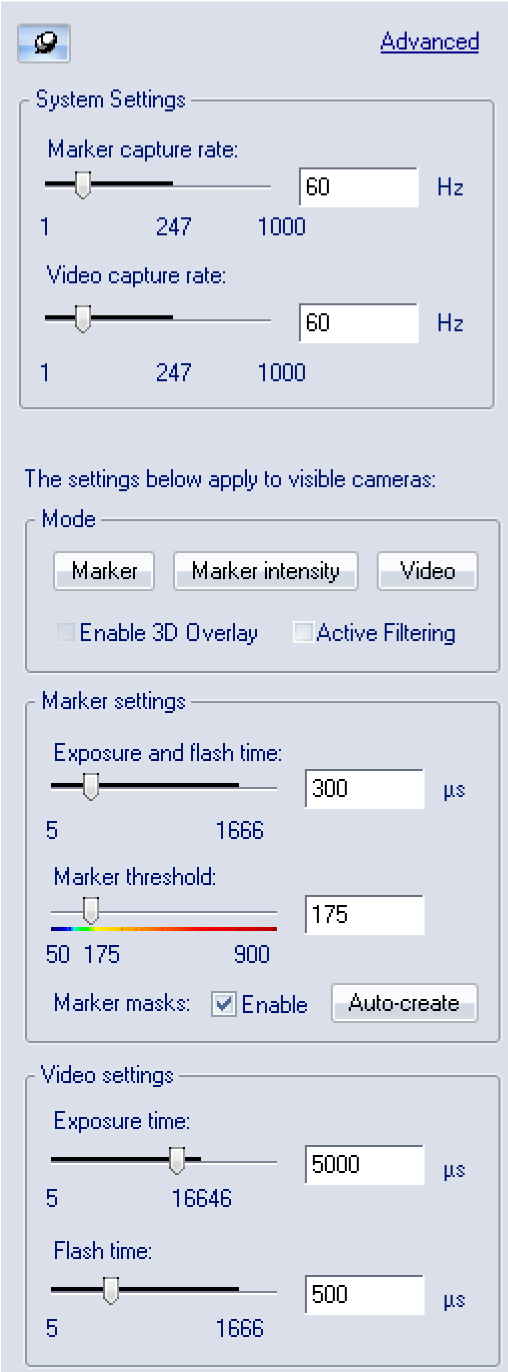
   **Zoom in/out the 2D view**
  Hold down both mouse buttons in the **2D view** window and move the mouse backward and forward, the view is zoomed out respectively in, in reference to the original position of the cursor. The zoom position is displayed in a view in the top right-corner of the camera.

  The mouse wheel can also be used to zoom. Click on the **Zoom** button to use the left mouse button for zoom.
  Note: With the mouse buttons the zoom is continuous, while with the mouse wheel it is done in steps.

   **Translate the 2D view**
  Translate can only be used if the 2D view is zoomed in. Hold down the right mouse button in the **2D view** window and move the mouse to translate. Click on the **Translate** button to use the left mouse button for translation.

Camera settings sidebar



The **Camera settings** sidebar contains the basic settings for the Oqus camera. Use these settings when setting up the camera system to get the best data. The sidebar slides out when you move the mouse on the edge to the right in the 2D view window. It can be pinned with the pin at the top so that it is always shown.

All of the settings, except for the **System settings**, applies only to the currently visible cameras in the 2D view window. Therefore it is important to use the buttons at the bottom of the 2D view window to select the cameras for which you want to change a setting.

Note: If the currently visible cameras have different settings it will say **Differs**. When changing such value it will set all the currently visible cameras to the same setting.

The following settings are available on the sidebar.

Advanced
Go to the **Advanced (Camera settings)** page in the **Project options** dialog, see chapter “Advanced (Oqus)” on page 143. The currently visible cameras will be selected in the camera list.

System settings
These settings applies to all cameras in the system.

Marker capture rate
The capture rate that is used by cameras measuring markers.
Note: If the camera system includes different Oqus types then the dark blue bar indicates the capture rate for the Oqus type with the lowest limit for the maximum image size.

Video capture rate
The capture rate that is used by cameras measuring video.
Note: If the camera system includes different Oqus types then the dark blue bar indicates the capture rate for the Oqus type with the highest limit for the maximum image size.

The capture rates can be increased above the limit for maximum image size. Then the image size is automatically reduced for all of the cameras in the system. The limit for the maximum image size is displayed with a dark blue bar and with a number below the slider. If the image size has been set manually for a camera the reduced image size will have the same relations for x and y.

Mode
These settings applies to all of the currently visible cameras in the 2D view window.

Marker, **Marker intensity** and **Video**
Switch between the different modes. The mode can also be changed individually on a camera from the **2D view window** menu.

Enable 3D overlay
Toggle the 3D overlay on and off. The 3D overlay can also be turned on individually for a camera from the **2D view window** menu.

Active Filtering
Enable the **Continuous** setting for the **Active filtering** mode, the cameras will capture two images to remove the background light. For information on how to use active filtering see chapter “How to use active filtering” on page 264.
Note: Only available for 3+-series and 1-series camera. However you can still turn on active filtering of those models, even if there are other camera models in the system.

Marker settings
These settings applies to all of the currently visible cameras in the 2D view window.

Exposure and flash time
The time used by the camera to capture the image in marker mode, for advice on this setting see chapter “Tips on marker settings in QTM” on page 250. The current maximum **Exposure and flash time** is displayed with a dark blue bar and a number below the slider.
Note: If the camera system includes different Oqus types, then the

maximum **Exposure and flash time** can differ depending on which cameras are displayed in the 2D view.

Marker threshold
The intensity level in the image used to detect markers, where the default value is 175. For example a lower value means that areas of less bright pixels will become markers, for advice on this setting see chapter "Tips on marker settings in QTM" on page 250.
Below the slider is the color scale which used for color-coding the video image in Marker intensity mode. The image will be green at the marker threshold and then blue below and yellow to red above threshold.

Marker masks
Setting for marker masks.

Enable
If deselected the marker masks in the current cameras will not be used by the camera. The masks will be grey and the markers below them will appear.

Auto-create
Create masks over all of the visible markers in the current cameras. It is important to make sure that it is only unwanted reflections that are visible when pressing the button. For more information about marker masking see chapter "How to use auto marker masking" on page 265.
Note: There can be 5 masks for each camera, so if there are more unwanted reflections than 5 the bottom ones will not be covered by masks.

Video settings
These settings applies to all of the currently visible cameras in the 2D view window.

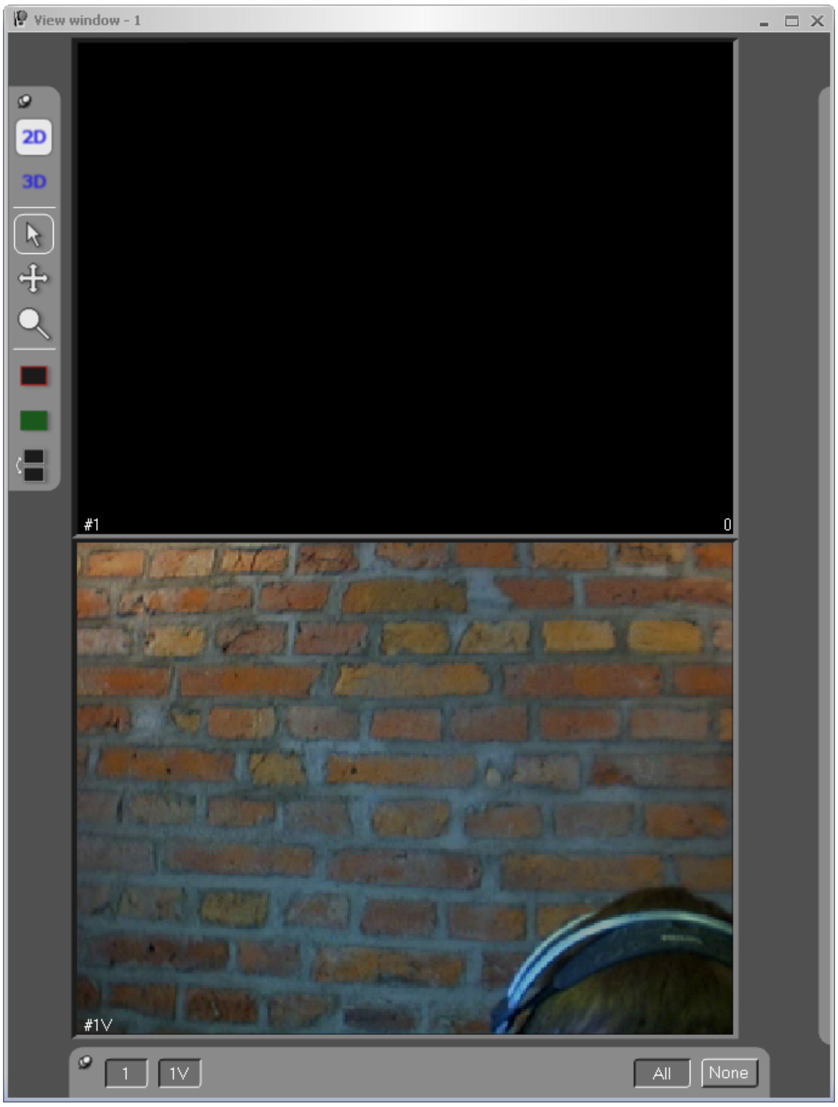
Exposure time
The time used by the cameras in video mode. Set it to a value where the image is bright enough, for more information see chapter "Outline of how to capture high-speed video" on page 258. The current maximum **Exposure time** is displayed with a dark blue bar and a number below the slider.
Note: If the camera system includes different Oqus types, then the maximum **Exposure time** can differ depending on which cameras are displayed in the 2D view.

Flash time
The time of the IR flash in video mode. This setting can be set to 5 microseconds unless you have markers placed on the subject that you want to be visible in the video. The current maximum **Flash time** is displayed with a dark blue bar and a number below the slider.

The settings are the same as on the **Camera settings** page in the **Project options** dialog, refer to chapters "Camera settings (Oqus)" on page 140 and "Advanced (Oqus)" on page 143 for more details about the settings.

DV/webcam video in 2D view

The DV/webcam video is displayed last in the 2D view window. The video data from a web camera can be used for documentation purposes, but it is not used in the calculation of trajectories.



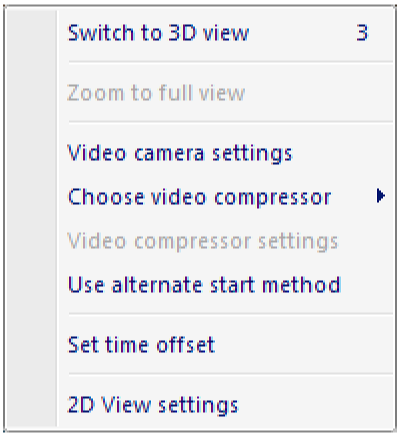
You need to save a default capture and file layout with a 2D view window to see the video automatically during the capture and in the file. To open a 2D view window with only the video cameras activated you can click on the **Video** button. Place the windows as you want them and then save the layout on the **Window** menu.

Note: You can have one video camera per 2D view window if you like. Use the camera buttons at the bottom of the 2D view window to select which cameras to view.

In preview mode the picture is the current view of the web camera. In file mode the picture is taken from the saved video file and it shows the video frame that corresponds to the capture frame. The video probably has fewer frames than the motion capture and therefore the numbers will not be the same. For information on how to record a video file see chapter “How to use DV/webcam video devices” on page 365.

In file mode the current frame number and time is displayed in the upper left corner, e.g. **Frame: 10** and **Time: 0.1 s**. This is probably not the same frame as the motion capture frame. The time does not have to be the same as the measurement time either, if the video offset has been changed, because the time that is displayed is the time from the start of the video file.

The settings for the video cameras are presented in the **View window** menu, which is opened by right-clicking on a video camera in the 2D view window. It is important to notice that the settings are individual per video camera so that you have to right-click in all of the video cameras if you want to change settings on several video cameras.



The following options are different from the options on the regular **2D view window** menu. The options are also different in RT/preview respectively File mode.

Video camera settings
These settings are specific for each type of DV/webcam, for more information see the manual of the web camera.
Note: Only available in preview mode.

Choose video compressor
Choose a video codec from a list of the installed codecs on the computer. The compressor is selected individually per DV/webcam. For more information see chapter "Compression of video from video cameras" on page 365.
Note: Only available in preview mode.

Video compressor settings
Open the video compressor settings for the currently selected codec of the camera.
Note: Only available in preview mode.

Set time offset ...
Set the time offset between the video file and marker data.

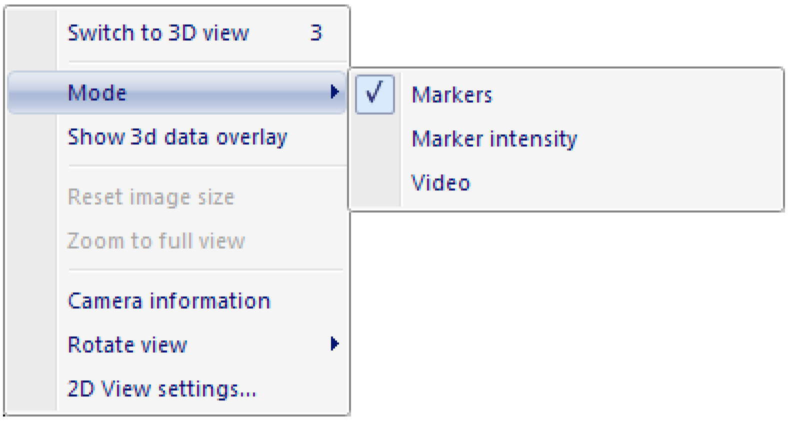
Use alternative start method
Use this alternative if the video capture does not start or does not start at the correct time.
Note: Only try this alternative if the default for some reason doesn't work.
Note: Only available in preview mode.

Remove this file link
Delete this video file from the QTM file. The AVI-file will not be deleted only the link in the QTM file. A video file can also be imported with **File/Import/Add link to video file**.
Note: Only available in file mode.

2D view window menu

The options for individual cameras are presented in the **2D view window** menu, which is opened by right-clicking in a 2D view. It can also be opened on the **View** menu. The menu differs between RT/preview mode and File mode. The image below is for a Oqus in RT mode.

Note: There are additional options when right-clicking on DV/webcam or a Point Grey camera, see chapter "DV/webcam video in 2D view" on page 42 and "Capturing video with Point Grey" on page 369.



The following options are available in the 2D view:

Switch to 3D view
Switch the View window to 3D view.

Mode (Oqus)

For a description of the different camera modes see chapter “Video preview in QTM” on page 256.
Note: This option is only available when QTM is in preview mode before you start a capture.

Markers
Switch to the default **Markers** mode.

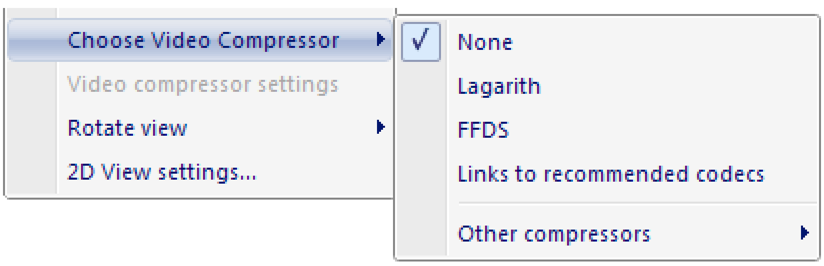
Marker intensity
Switch to the **Marker intensity** mode.

Video
Switch to the **Video** mode.

Show 3D data overlay
Overlay the 3D data on the 2D view. This can be done on any camera independent of whether it is **Marker** or **Video** mode. For more information see chapter “3D data overlay” on page 47.

Reset image size (Oqus)
Reset to full image size for a camera with reduced image size.
Note: This option is only available when QTM is in preview mode before you start a capture.

Zoom to full view
Reset the zoom so that the whole image is shown for the camera. This is the same as double-clicking on the miniature in the top right corner.



Choose video compressor (Oqus)
Choose a video codec from a list of the installed codecs on the computer. The compressor is selected for all Oqus cameras in Video mode. For more information

see chapter “Video compression”.

Note: Only available if the Oqus camera is Video preview mode.

Video compressor settings (Oqus)
Open the video compressor settings for the currently selected codec of the camera.

Note: Only available if the Oqus camera is Video preview mode.

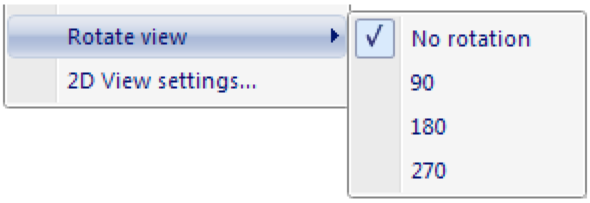
Delete 2D data (Oqus)
Delete 2D data imported from QVA.

Camera information (Oqus)
Show the camera information in a dialog, i.e. Oqus type, IP address and so on.

Note: This option is only available when QTM is in preview mode before you start a capture.

Review camera settings
Show the settings used for the camera in the current file, for more information see chapter “Review camera settings” on the facing page.

Note: This option is only available when QTM is in file mode.



Rotate view
Change the rotation of the displayed 2D view so that it matches the camera rotation. For example if the camera is placed upside down you can use the 180 degree rotation.

The rotation is stored with the file, so you can rotate the cameras after you have calibrated the camera system. The 2D view rotations will then be stored in the QTM file. It is also possible to rotate the 2D views in a QTM file, but to save it you must make sure to make another modification to the file as well.

2D View settings...
Open the 2D view settings page in the **Project options** dialog so that you can change the 2D view settings, for example the 3D overlay settings. For more information see chapter “2D view settings” on page 220.

Preview Mode (ProReflex)

Note: This option is only available when QTM is in preview mode before you start a capture.

> **Markers**
> Switch to the default **Markers** mode.
>
> **Viewfinder**
> **Switch to the Viewfinder mode.**
>
> Note: Only one camera at a time can be set to the **Viewfinder** mode.

Viewfinder settings
See chapter “Viewfinder” on next page for more information on **Viewfinder** and its settings.

3D data overlay

The 3D data can be overlayed on the Oqus video data to show the 3D view from the camera viewpoint. This can for example be used for showing the force arrow in the video of someone stepping on a force plate or to check which 2D markers that are used for the 3D calculation.

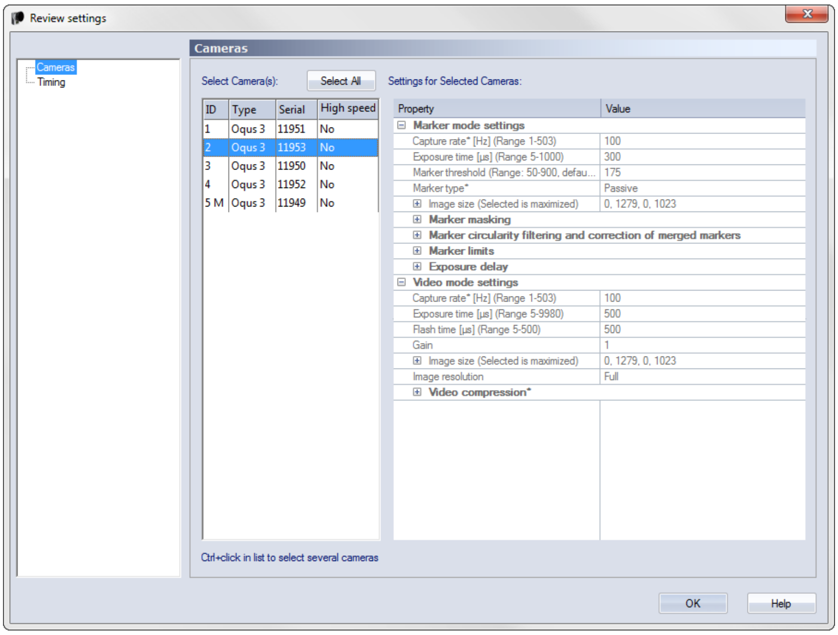
Follow these steps to activate the 3D data overlay.

- Calibrate the camera system, including the Oqus cameras that are used in video mode.
- Open a **2D view** window in RT/preview or a file.
- Right-click on the camera where you want to turn on the overlay and select **Show 3D data overlay**.
  Note: The 3D data can also be overlayed on the 2D data, for example to check how much of the measurement volume that is seen by the camera.
- The 3D elements displayed in the overlay and the opacity can be changed on the **2D view settings** page in the **Project options** dialog, see chapter “2D view settings” on page 220.

  The marker and video data display in the 2D view can be switched between linearized and unlinearized on the **2D view settings** page. To match the 2D data with the 3D data the data must be linearized.

Review camera settings

To review the camera settings in a file right-click on a camera 2D view and the select the **Review camera settings** option. It will open the ... dialog, which displays the same settings as on the **Advanced Camera settings** page and **Advanced Timing** page in the **Project options** dialog.



The settings are greyed out so that they cannot be changed, but otherwise you can navigate in the same way as in the **Project options** dialog. Any setting that was not available in the QTM version in which the file was capture will be set to its default value. For more information about the settings see its respective chapter in “Advanced (Oqus)” on page 143 and “Advanced (Timing)” on page 132.

Viewfinder

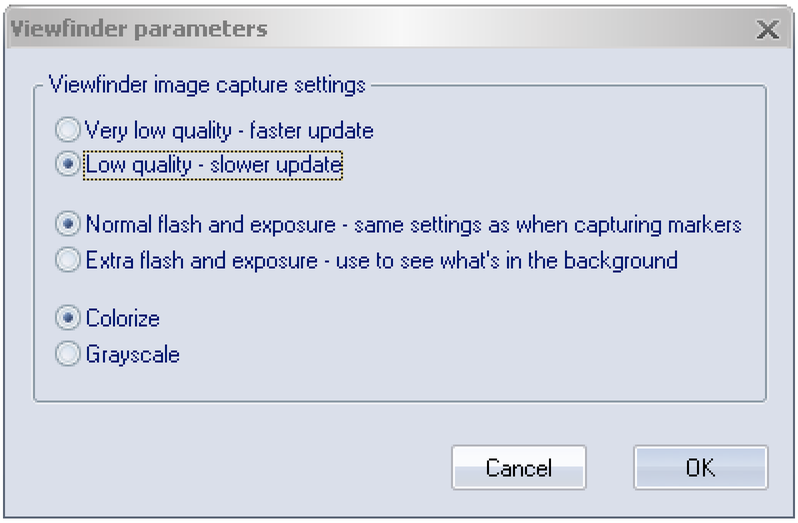
This function is only for ProReflex cameras, for the video preview in Oqus cameras see chapter “Video preview in QTM” on page 256.

With the help of the **Viewfinder** mode in the **2D view** window it is possible to look at a greyscale image of a camera view. The mode can for example be used to see the field of view of the camera and to detect any unwanted reflections. It can only be used in preview mode before you start a capture and it is not possible to record the greyscale image in QTM. The greyscale image may seem to have a low quality, this is due to the fact that the ProReflex MCU was not designed for this feature and several compromises have been made to get the greyscale image. One cause to the low quality is the IR-filter that deletes the visible light.

To activate the **Viewfinder** mode on a camera, right-click in the camera's **2D view** window to open the **2D view window** menu. Then click on **Preview mode** and then **Viewfinder**. The camera will switch to **Viewfinder** mode. This may take some time depending on the **Viewfinder settings**. The update frequency can be lower than 1 Hz for some settings. For information about the settings see chapter “Viewfinder parameters” below.

Note: Only one camera at a time can be set to **Viewfinder** mode. This means that when you switch to **Viewfinder** mode for one camera all of the other cameras will be set to **Markers** mode.

Viewfinder parameters



With the settings in the **Viewfinder parameters** dialog you can change the following parameters for the **Viewfinder** mode:

Image quality

Very low quality
The quality of the image is very low but the update frequency is higher, which means that you can see a moving object.

Low quality
The quality of the image is higher than for **Very low quality** and the update frequency is lower, which means that you can see more details but it is difficult to view a moving object.

Flash and exposure time

Normal flash and exposure
The flash and exposure are the same as in the **Markers** mode. The image will be very dark and you cannot see much more than the markers. However if you use the **Colorize** option you can see the trigger level where the camera detects a marker, see below.

Extra flash and exposure
The flash and exposure are much longer than the normal flash and exposure. The image is brighter and you will see more of the background. However how much that can be seen depends a lot on the aperture and the distance to the objects. With a small aperture and a distance of a couple of meters it is still not much more than the markers that can be seen.

Color or Greyscale

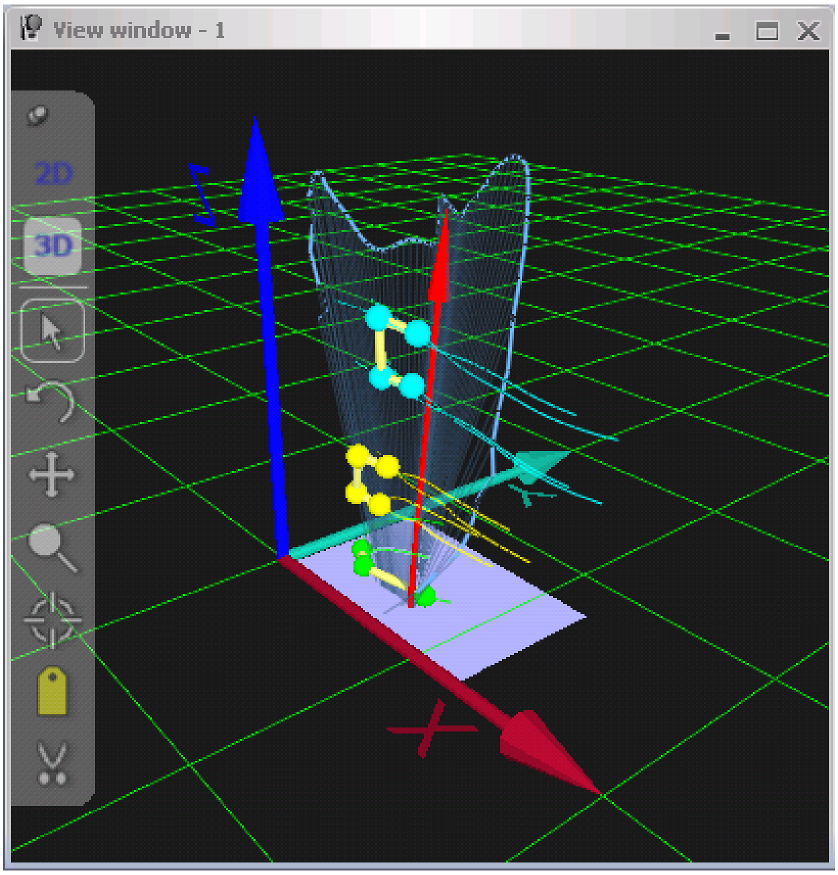
Colorize
The image is colorized depending on the brightness. With this option it is easier to see details than with **Greyscale**.

- For **Normal flash and exposure** the dark areas are black and it then follows the color chart up to red for the bright areas. The trigger level where the camera detects a marker is in the green color area according to the picture below. This means that if something in the image that is not a marker is green or above it will be detected as a marker by the camera.

- For **Extra flash and exposure** the dark areas are black and the next color is green, which means that the background will have a green color. There is no color which represents the trigger level since the flash and exposure has been altered.

Greyscale
The image is displayed in greyscale, this is the original format of the image.

3D view window



In 3D viewwindows the motion capture data is shown in 3D using the coordinate system that was defined in the calibration. How to use the 3D view is described in the chapter “Using and modifying the 3D view” on the facing page.

The 3D view window includes the following controls:

3D view toolbar
Contains tools for the 3D view, see chapter “3D view toolbar” on page 52.

3D view window menu
Options for the 3D view, see chapter “3D view window menu” on page 60.

The following graphics can be shown in a 3D view window:

- Axes of the coordinate system of the motion capture (X = red, Y = light green and Z = dark blue).
- A green grid, which is the floor of the measurement volume (e.g. Z = 0, if Z is the vertical axis of the measurement setup).
- The position and orientation of each camera in this specific setup.
- Text labels for some of the graphics.
- Markers for the current position of the trajectories, see chapter “Trajectories in 3D views” on page 54.
- Traces of the trajectories.
  Note: Only available in file mode.
- Bones between markers, see chapter “Bones in 3D views” on page 56.
- 6DOF bodies, see chapter “6DOF bodies in 3D views” on page 55.

- Purple squares, which display the location of the force plates.
- Red force vectors displaying the current force on the force plate, see chapter “Viewing force data” on page 311.
- A blue force trace displaying the force applied to the force plate during the measurement.
  Note: Only available in file mode.
- Camera view cones, see chapter “Camera view cones in 3D views” on page 56.
- Covered and calibrated volumes, see chapter “Volumes in 3D views” on page 57.
- A white outlined box for the bounding box used by the 3D tracker, see chapter “Bounding box that restricts the data” on page 174.

All of these graphics can be toggled on and off and changed on the **3D view settings** page in **Project options** dialog, see chapter “3D view settings” on page 221.

Using and modifying the 3D view

The following chapters describe the actions that can be performed on objects in the 3D view and to the 3D view itself.

The **Selection** cursor is the default cursor. With modifiers it can be used to perform any of the tasks in the 3D view. The tasks include everything from selecting and identifying trajectories to changing the view of the 3D view.

For a description of the other cursors see chapter “3D view toolbar” on next page.

For a description how to change the 3D view, see chapter “Change the viewpoint of the 3D view” below.

For a description of the different actions that can be performed on the objects in the 3D view see chapters:

“Trajectories in 3D views” on page 54

“6DOF bodies in 3D views” on page 55

“Bones in 3D views” on page 56

“Camera view cones in 3D views” on page 56

“Volumes in 3D views” on page 57

“Other objects in 3D views” on page 60

Change the viewpoint of the 3D view

The 3D view can be rotated, translated or zoomed so that the data can be seen from the viewpoint that you want. For most actions which change the view of the 3D view a red crosshair is shown in the middle of the 3D view. You can always reset the rotation to the default view by double-clicking anywhere in the background.

The following actions can be performed in 3D viewwindows to change the view of the 3D view:

Rotate the 3D view
Hold down the left mouse button in the 3Dview window and move the mouse, the view is rotated freely around the red crosshair. Hold down Ctrl + Shift to limit the rotation to the current plane of view.

Translate the 3D view
Hold down the right mouse button in the 3D viewwindow and move the mouse,

the focus is moved in the 3D view.
Another way to move the focus is to hold down Shift+C and click on a trajectory then the focus is moved to that marker.

Zoom in/out the 3D view
Hold down both mouse buttons in the 3Dview window and move the mouse backward and forward. The view is zoomed out and in respectively, in reference to the position of the red crosshair. The mouse wheel can also be used to zoom. You can also zoom to the current position of the mouse cursor by holding down Ctrl.

Note: With the mouse buttons the zoom is continuous, while with the mouse wheel it is done in steps.

3D view toolbar

The **3D view** toolbar contains tools for manipulating the 3D view and the trajectories and for switching between 2D and 3D view. From top to bottom the icons have the following use.

2D button
Switch to 2D.

3D button
Switch to 3D.

Selection cursor
Use the normal mouse behavior. It has the following keyboard modifiers:

Shift - Add trajectories to the selection.

Ctrl - Add/Remove trajectories to the selection.

Shift + drag - Select trajectories with an area. If there are traces below the area these trajectories will be selected as well.

Alt - Select a trace.

Alt + Shift - Add trace to the selection.

Rotate cursor
Use the left mouse button to rotate the 2D view. The difference to the **Selection** cursor is that you can no longer select anything with this cursor. It has the following keyboard modifier:

Ctrl + Shift - Rotate in the plane of the current view. It is possible to get strange rotations with this modifier, use double-click to reset to default rotation.

Translate cursor
Use the left mouse button to translate the 2D view. This is the same alternative as using the right mouse button with the **Selection** cursor.

Zoom cursor
Use the left mouse button to zoom the 2D view. This is the same alternative as using both mouse buttons or the mouse wheel with the **Selection** cursor. It has the following modifier:

Ctrl - Use the alternative zoom method. For the default settings the alternative method is zoom to the current position of the cursor.

Center trajectory cursor
The view is centered on the trajectory that you select. This is the same as using the Shift + 'c' key with the **Selection** cursor.

Quick identification cursor
Identify the trajectory that you select as the selected label in the **Labeled trajectories** window, for more information see chapter “Manual identification of trajectories” on page 300. This is the same as holding down Ctrl + Alt with the **Selection** cursor. It has the following modifier:

Shift - Join the trajectory to the previous label in the **Labeled trajectories** window. This option can for example be used if the last identified trajectory only covers a part of the measurement range, then you can add the rest by holding down Shift when clicking on the next part.

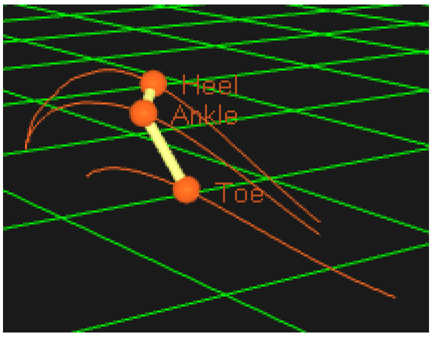
Cut trajectory trace cursor
Click on a trace to cut it in two parts. This is the same as using the Shift + 'x' key with the **Selection** cursor.

Note: The gap between the two parts is there to visualize that there are two parts it is not a missing frame.

Camera view cones menu
Click on the icon to open the **Camera view cones** menu, for more information see chapter “Camera view cones in 3D views” on page 56.

Volumes menu
Click on the icon to open the **Volumes** menu, for more information see chapter “Volumes in 3D views” on page 57.

Trajectories in 3D views



The trajectories of a measurement are shown with markers on their current position in the 3D view, where the colors of the trajectories are the same as in the **Trajectory info** windows.

In file mode, the position throughout the measurement can be displayed with a trace, where the length of the traces is controlled by the bottom sliders on the **Timeline control** bar. The trace of a trajectory is thicker when the focus is set on its trajectory in the **Trajectory** window, i.e. there is a dashed box around it.

You can change settings for the marker and trace display on the **3D view settings** page in the **Project options** dialog. Among other things you can turn on the display of the marker labels.

The trajectories can either be created with 3D or 2D tracking, but the display in the **3D view** window is the same except that 2D tracked trajectories are displayed in a plane. The data of the marker is shown in a **Trajectory info** window, see chapter “Trajectory info windows” on page 65.

The following actions can be performed with the **Selection** cursor on a trajectory in the 3D view.

Select
Click on the marker or the trace of the trajectory to select it. Use the following keys to modify the action when you click on a trajectory.

Hold down Alt to only select a part of the trajectory.

Use Shift and Ctrl to add and delete (only Ctrl) trajectories to the selection.

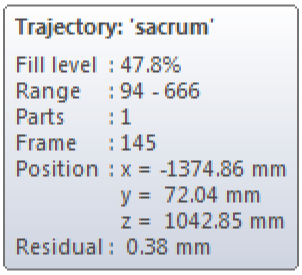
Hold down Shift and then click and drag the mouse to use area selection.

Identification
Trajectories can be drag and dropped from the 3D view to any of the trajectories window to change the identity. For example drop an unidentified trajectory on an empty label in the **Labeled trajectories** window to identify it.

A trajectory can also be dropped on the trace of another trajectory to join them. This requires that no parts of the trajectories completely overlap each other.

Trajectory info window menu
Right-click on a trajectory to open the **Trajectory info window** menu for the selected trajectories, see chapter “Trajectory info window menu” on page 69.

Information
Place the cursor over a marker or a trace to see a tool-tip with information about the trajectory.

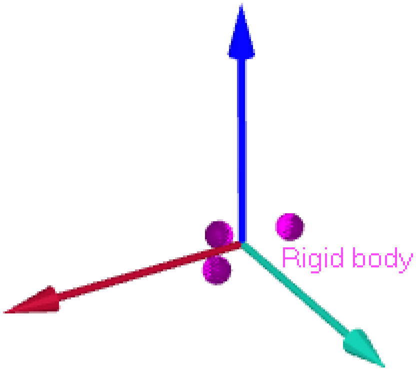


Delete

Use the Delete key to delete selected trajectories and parts of trajectories directly in the **3D view** window.

Note:Trajectories in the **Discarded** trajectories window are hidden by default.

6DOF bodies in 3D views



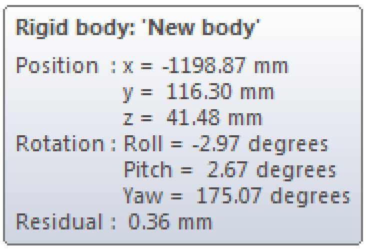
Measured 6DOF bodies are displayed as separate local coordinate systems in a **3D view** window, where the origin is placed at the position of the local origin of the 6DOF body. The axes of the local coordinate system have the same color codes as the coordinate system of the motion capture.

Virtual markers corresponding to the points of the 6DOF body definition are also displayed for each 6DOF body. The markers will therefore be displayed with two colors in the 3D view. The colors of the rigid body are set for each 6DOF body definition on the **6DOF tracking**page in the **Project option** dialog. The actual trajectories will automatically get a slightly brighter color.

You can change settings for the 6DOF body display on the **3D view settings** page in the **Project options** dialog.

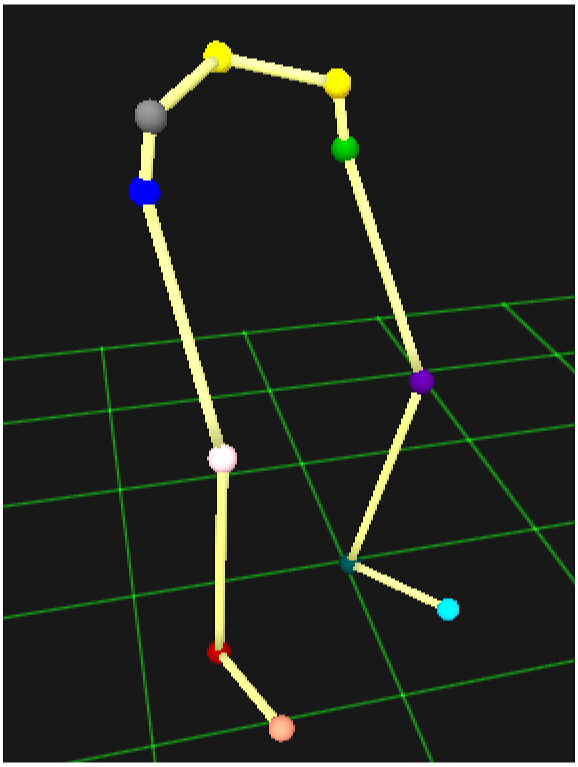
The 6DOF data can be viewed in the **Data info** window or exported to a TSV file and Matlab, see chapter “6DOF data information” on page 81 respectively chapters “Export to TSV format” on page 314 and “Export directly into Matlab” on page 326.

Place the mouse over the local coordinate system to see information about the body in the current frame.



Note: In a capture file, definition, name and color of a 6DOF body can only be changed by reprocessing the file, see chapter “Reprocessing a file” on page 281.

Bones in 3D views



Bones are used to visualize the connection between two markers in 3D views, e.g. if the measurement is done on a leg, bones can connect hip to knee, knee to foot and so on. The bones settings are set on the **3D view settings** page in the **Project options** dialog.

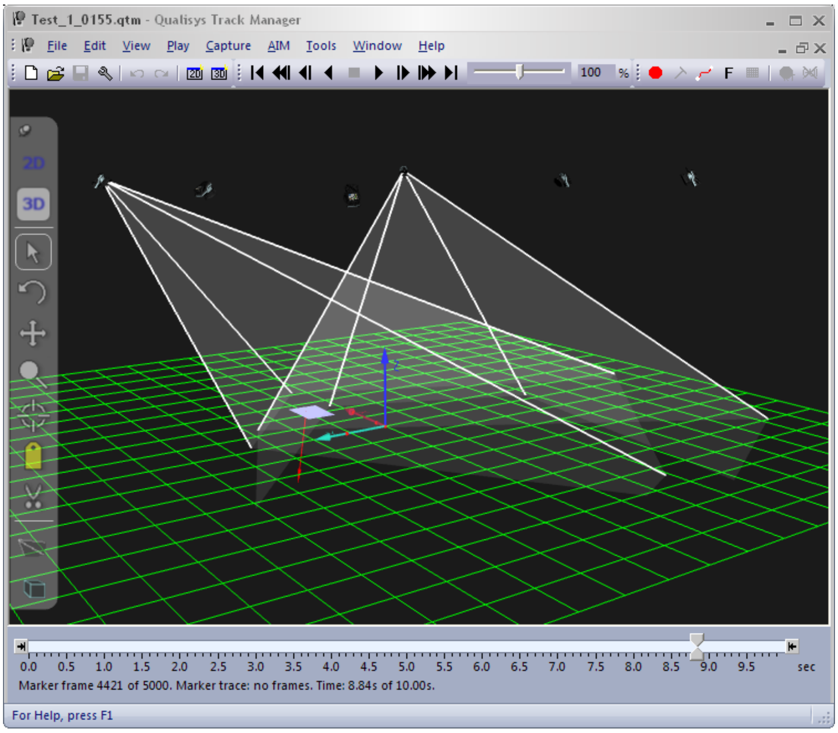
To create a bone, hold down Shift and select a pair of labeled trajectories by clicking in the 3D view window. Then press 'b' with an active 3D view window or click **Bone** in the **3D view window** menu and then **Create bone** to create the bone. Several bones can be created at the same time if the trajectories are selected one at a time with Shift + Left-click in the 3D view window. The bones will then be created between the bones in that order.

To delete one or more bones, select the bones. Then press delete or right-click on the bone and click **Delete bone**.

Note: If bones are included in the capture file that is used to generate an **AIM** model, the bones will be created again when the **AIM** model is applied. The bones are also saved in label lists, so that they are loaded if a label list is loaded.

Camera view cones in 3D views

The camera view cones display the field of view of the camera, i.e. what a camera will be able to measure. It can be used to evaluate how the cameras are placed in the system so that the placement can be better optimized. By enabling multiple cones, you can also study what is covered by a certain subset of the camera system.



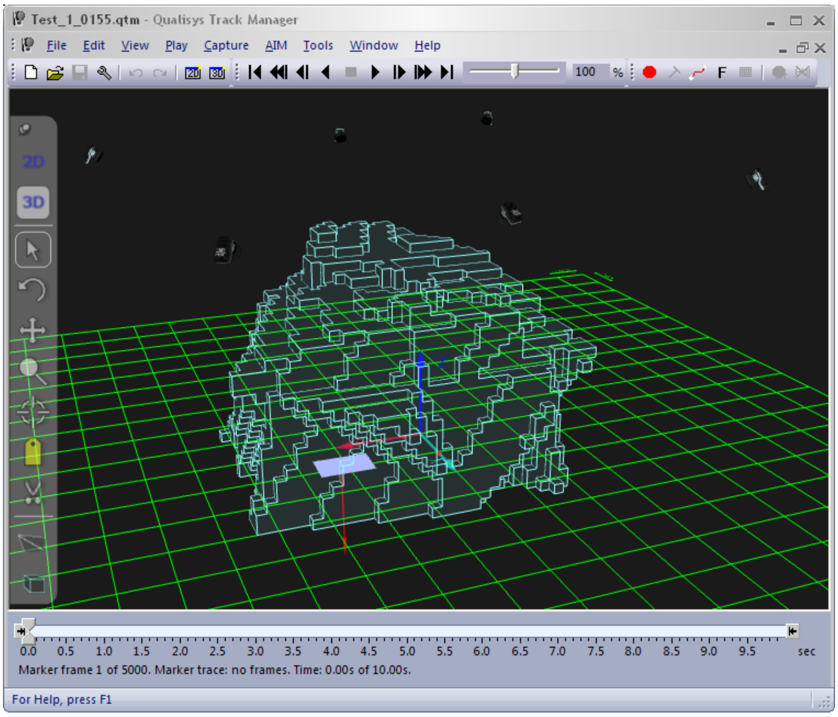
The view cones can be enabled per camera from the **Camera view cones** menu on the **3D view** toolbar. Click on the **Camera view cones** button  to open the dialog and enable the cones with the **Show camera view cones** checkbox. The feature can also be enabled on the **3D view settings** page in the **Project options** dialog.

The length of the cones is determined by the **Smallest marker size visible** setting, which defines the marker size that should be visible in the entire cone. The cones are by default cut at the floor level, but that can be changed by disabling the **Cut covered volume at floor level** option on the **3D view settings** page in the **Project options** dialog.

Select which camera cones that are to be shown with the **All**/**None** buttons and camera check boxes.

Volumes in 3D views

QTM can help you see the volume in which you will be able to measure by calculating the covered and the calibrated volumes. The view cones can also be used for visualizing the FOV, see chapter “Camera view cones in 3D views” on previous page.

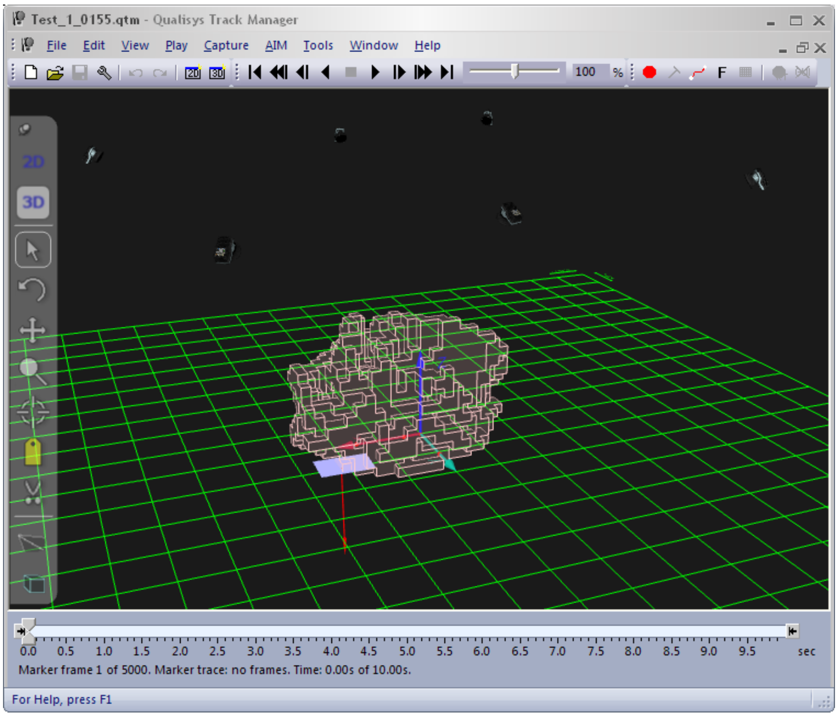


The covered volume is displayed as light blue cubes. It is the volume that is seen by a certain number of cameras, specified by the user with the **Cameras required to consider volume covered** setting on the **3D view settings** page in the **Project options** dialog. The covered volume can be used to determine where 3D data can be measured and is calculated by combining the view cones and is therefore affected by the length of them, i.e. the **Smallest marker size visible** setting.

The default marker size differs between the camera models. e.g. for Oqus 3-series the default marker size is 12 mm. The covered volume is also cut default by the floor level, but that can be changed by disabling the **Cut covered volume at floor level** option on the **3D view settings** page in the **Project options** dialog.

The default required number of cameras is three cameras for calculating the covered volume, which most of the time gives the most likely covered volume. If you use only two cameras to calculate the volume there will be some parts that are actually very difficult to reach with the wand.

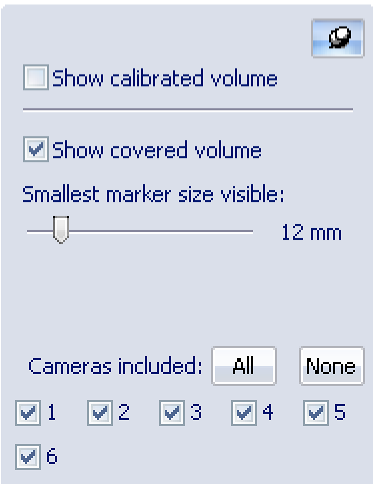
It is important to notice that the covered volume does not consider whether it is likely that markers are occluded by the subject or not. To simulate this you can use the camera selection in the **Volumes** menu see below.



The calibrated volume is displayed as light red cubes. It is the volume that the wand moved through when the camera system was calibrated. It therefore indicates where the most accurate 3D results can be expected and can be used to evaluate if the wand needs to be moved in a larger volume. 3D data will however be calculated outside the calibrated volume as well. This is usually not a problem as long as the markers are within a few decimeters of the calibrated volume. Use the 3D residual to evaluate if the data is good enough.

Note: The calibrated volume can only be displayed in files calibrated with a calibration file which is processed in QTM 2.3 or later.

The volumes can be enabled from the **Volumes** menu on the **3D view** toolbar. Click on the **Volumes** button  to open the dialog and enable the volumes with the **Show calibrated volume** checkbox and the **Show covered volume** checkbox. The features can also be enabled on the **3D view settings** page in the **Project options** dialog.



The maximum distance from the cameras is determined by the **Smallest marker size visible** setting, which defines the marker size that should be visible in the entire volume.

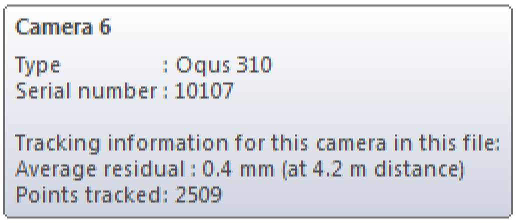
Select which cameras that are used in the calculation with the **All**/**None** buttons and camera check boxes. By choosing which cameras are considered when creating the covered volume, you can for example determine what happens to the volume when one camera is occluded. Another case that can be simulated is when the cameras are mounted on two sides and can only see markers on one side of the subject, e.g. on a human walking through the volume. Then you can turn off all of the cameras on one of the sides to see the volume where the subject can be viewed by the rest of the cameras.

Other objects in 3D views

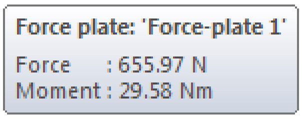
The display of other objects (axes, grid and so on) in the 3D view can be changed from the **3D view settings** page in the **Project options** dialog.

Place the mouse over the following objects to see information about the object.

Camera
Show camera information. The tracking information is the same as in the **File information** dialog, see chapter “3D tracking test” on page 284.

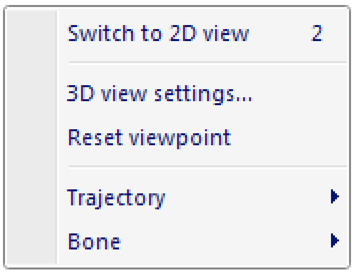


Force plate or **Force arrow**
Display the current force data.



3D view window menu

The **View window** menu is opened by right-clicking in a 3D view window. It can also be opened on the **View** menu. Note that if you right-click on bones or markers in the 3D view then the **Bone** menu respectively **Trajectory info window** menu is opened instead.



In a **3D view** window the following actions can be performed:

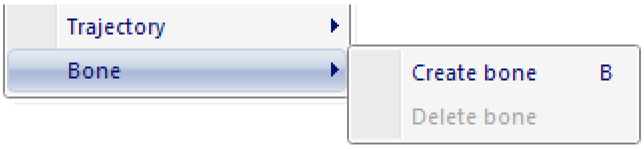
Switch to 2D view
Switch the View window to 2D view.

3D view settings...
Open the 3D view settings page in the **Project options** dialog so that you can change the display options for the objects in the 3D view. For more information see chapter “3D view settings” on page 221.

Reset viewpoint
Reset the camera viewpoint to the default, which is centered on the coordinate system.

Trajectory
Open the **Trajectory info window** menu for the selected markers, see chapter “Trajectory info window menu” on page 69.

Bone
For more information on bones see chapter “Bones in 3D views” on page 56.

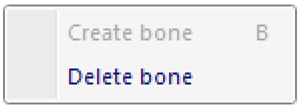


Create bone
Create a new bone between two selected trajectories.

Delete bone
Delete the selected bones.

Bone menu

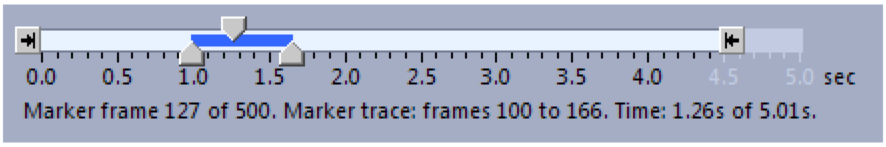
The **Bone** menu is opened when right-clicking on a bone, but it can also be opened on the **Edit** menu and in the **3D view window** menu.



Create bone
Create a new bone between two selected trajectories.

Delete bone
Delete the selected bones.

Timeline control bar



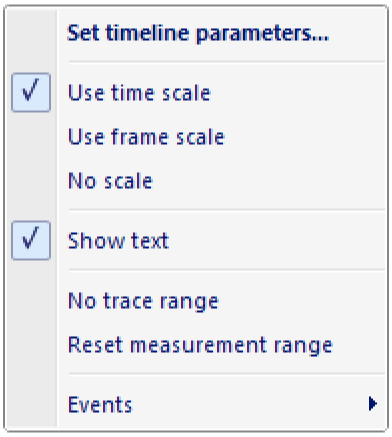
The **Timeline control** bar is shown at the bottom of the QTM window in file mode. It is used to indicate and to select current frame, trace range, measurement range and events.

The current frame is indicated by the top slider, the exact frame is shown in the text below the control bar. To go to a frame, left-click on the desired position on the timeline. You can also drag the top slider by left-clicking on the slider and hold down the button. Then move the mouse to the left or to the right.

The trace range is the amount of trace that is shown in the **3D view** window and it is selected with the two bottom sliders. Drag the sliders to the desired positions for the trace range, the exact position of the sliders is shown in the **Status** bar. The sliders are only visible for 3D data in **3D view** windows. If they are not visible, right-click in the **3D view** window and click **Options** and then **Display traces**.

The measurement range is the amount of the original measurement that is used in the analysis, i.e. when plotting, exporting or playing the data. It is selected with the two scroll boxes at the ends of the time range. Drag the boxes to the desired positions for the measurement range. The exact measurement range can be found in the **Timeline parameters** dialog, just double-click on the **Timeline control** bar to open the dialog.

The events are displayed in the timeline as red triangles ▼ above the timeline. Place the mouse over the event on the timeline to see information about the event. Right-click on an event to open the **Events** menu, see chapter “Events menu” on the facing page. For more information about how to use events see chapter “Events” on page 309.



Use the **Timeline** menu to modify the **Timeline control** bar.

Set timeline parameters
Set the parameters of the **Timeline control** bar, see chapter “Timeline parameters” on the facing page. The dialog can also be opened by double-clicking on the **Timeline control** bar.

Use time scale
Show time scale in seconds in the timeline.
Note: Not available with SMPTE timestamp.

Use frame scale
Show marker frames in the timeline.

No scale
Do not display any scale.

Show text
Toggle the display of information about **Marker frames**, **Marker trace** and **Video frames** and **Time/SMPTE timestamp** in the timeline.
Note: When there are both marker and video data from Oqus cameras in the file both the **Marker frame** and the **Video frame** are displayed.
Note: The time displayed is either the measurement time starting at 0 seconds or the SMTPE timestamp if that functionality is activated. The SMPTE timestamp is at 30 Hz, which means that there can be several frames per timestamp and is indicated by the last number in the timestamp.

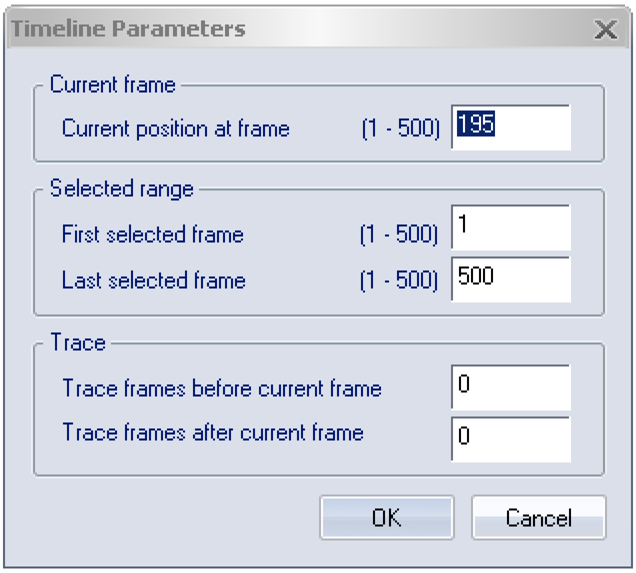
No trace range
Reset the trace range to no trace.

Reset measurement range
Reset the measurement range so that it contains the whole measurement.

Events
Open the **Events** menu, see chapter “Events menu” on the facing page.

Timeline parameters

In the **Timeline parameters** dialog you can set the parameters of the **Timeline control** bar.



The parameters in the dialog are the same which can be set manually in the **Timeline control** bar. The numbers inside the parenthesis show the possible values of that parameter. When setting the different parameters you will be warned if the parameter is outside its possible range. The parameters are as follows:

Current position at frame
The current frame of the measurement, indicated by the top slider.

First selected frame
The first frame in the measurement range, indicated by the scroll box.

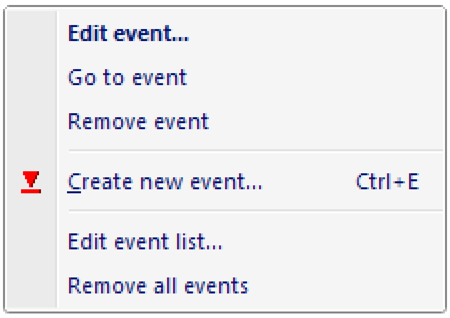
Last selected frame
The last frame in the measurement range, indicated by the scroll box.

Trace frames before current frame
The start of the trace range, indicated by the left bottom slider.

Trace frames after current frame
The end of the trace range, indicated by the right bottom slider.

Note: The trace parameters can be negative, which then means that the trace can start after the current position or stop before the current position.

Events menu



The **Events** menu is opened by right-clicking on an event in the **Timeline control** bar. The Events menu is also available on the **Timeline** menu and **Edit** menu, but then the actions are limited to the last three.

Edit event
Edit the current event. It will open the **Edit Event** dialog, where you can edit **Label**, **Time** and **Frame**.

Go to event
Move the current frame of the measurement to the current event.

Remove event
Remove the current event.

Create new event
Open the **Add event** dialog to create new event on the current frame, for more information see chapter “Adding events” on page 309.

Edit event list
Open the **Edit event list** dialog to edit all of the events in the current file, for more information see chapter “Viewing and editing events” on page 310.

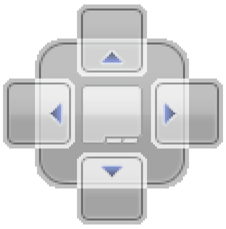
Remove all events
Remove all events in the current file.

Trajectory info windows

In the **Trajectory info** windows the trajectories of the measurement are listed. The trajectories can either be created with 3D or 2D tracking, but they are handled in the same way in the **Trajectory info** windows. The following three **Trajectory info** windows are used in QTM:

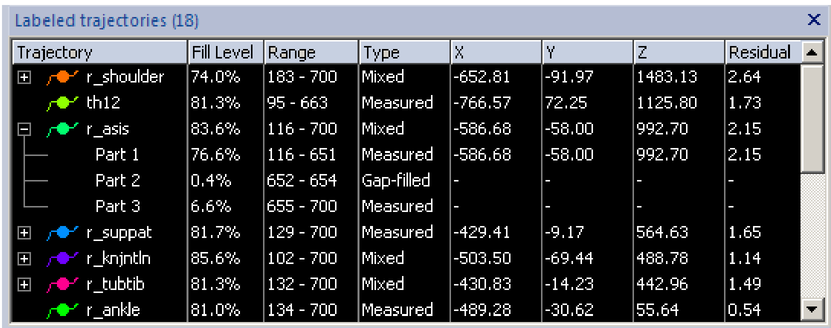
- **Labeled trajectories** window
  Contains the trajectories that have been identified (labeled).
- **Unidentified trajectories** window
  Contains unidentified trajectories.
  Note: After a motion capture the trajectories are placed by default in the **Unidentified trajectories** window, as long as identification is not performed in the automatic processing.
- **Discarded trajectories** window
  Contains trajectories that have been manually deleted.

The windows are tool windows and the **Labeled** and **Unidentified trajectories** window can be opened in preview mode. By default the **Trajectory info** windows are placed on the right side of the main window. However they can be moved to any position and then there smart docking markers to help you place the window, see below. If the windows are placed on top of each other they are then activated with the tabs at the bottom of the window.



The options for the trajectories are found on the **Trajectory info window** menu, see chapter "Trajectory info window menu" on page 69. It is opened by right-clicking on a trajectory or a group of selected trajectories.

Data in Trajectory info windows


Labeled trajectories (18)

Trajectory	Fill Level	Range	Type	X	Y	Z	Residual
r_shoulder	74.0%	183 - 700	Mixed	-652.81	-91.97	1483.13	2.64
th12	81.3%	95 - 663	Measured	-766.57	72.25	1125.80	1.73
r_asis	83.6%	116 - 700	Mixed	-586.68	-58.00	992.70	2.15
Part 1	76.6%	116 - 651	Measured	-586.68	-58.00	992.70	2.15
Part 2	0.4%	652 - 654	Gap-filled	-	-	-	-
Part 3	6.6%	655 - 700	Measured	-	-	-	-
r_suppat	81.7%	129 - 700	Measured	-429.41	-9.17	564.63	1.65
r_knjntln	85.6%	102 - 700	Mixed	-503.50	-69.44	488.78	1.14
r_tubtib	81.3%	132 - 700	Mixed	-430.83	-14.23	442.96	1.49
r_ankle	81.0%	134 - 700	Measured	-489.28	-30.62	55.64	0.54

The following data is listed in all of the **Trajectory info** windows:

Trajectory
The label of the trajectory and the color of its marker in 3D views. In addition to the color the symbol next to the label also shows if the trajectory and its trace are displayed in 3D views. The following symbols are used:

Both trajectory and trace are displayed.
Just the trajectory is displayed.

If a trajectory consists of more than one part there is plus sign (+) in front of the label. The parts are shown by clicking the plus sign or by using the left and right arrows.

Note: In the **Labeled trajectories** window the label can be edited by double-clicking on the label. However, in the other two windows the labels are always **Unidentified** and **Discarded** followed by a sequential number, which can be used to separate the trajectories.

Type
The type of trajectory, which can be one of the following types.

Measured
A trajectory or a part that has been tracked from the measurement data.

Mixed
A trajectory with a mix of **Measured**, **Gap-filled** and **Virtual** parts.

Gap-filled
A trajectory or part that has been calculated with the gap fill function.

Virtual
A trajectory or part that has been calculated from 6DOF body.

Fill level
The percentage of the current measurement range that the trajectory or part is visible. The measurement range is selected using the **Timeline control** bar.

Range
The range of frames, within the measurement range, with data for the trajectory or part.

X, Y and Z
The position (in mm) of the trajectory in the current frame. The coordinates use the coordinate system of the motion capture system set up when the system was calibrated.

Residual
The average of the different residuals of the 3D point. This is a quality check of the point's measured position.
Note: The residual is not available for 2D tracked trajectories.
Note: The residual is 0 for virtual and gap-filled trajectories.

Note: The number after the window name is the number of trajectories in the window.

Select and move trajectories

Select a trajectory in the **Trajectory info** window by clicking on the trajectory. It is possible to select multiple trajectories with standard Windows multiple selection methods, i.e. Ctrl+Left-click, Shift+Left-click and Ctrl+A (select all).

To deselect all trajectories, either click on the **Trajectory** heading to deselect all or use Ctrl+Left-click on the selected trajectories. The keyboard can also be used to select and deselect trajectories, use Ctrl+Space to select a trajectory and the up and down arrows to move in the list.

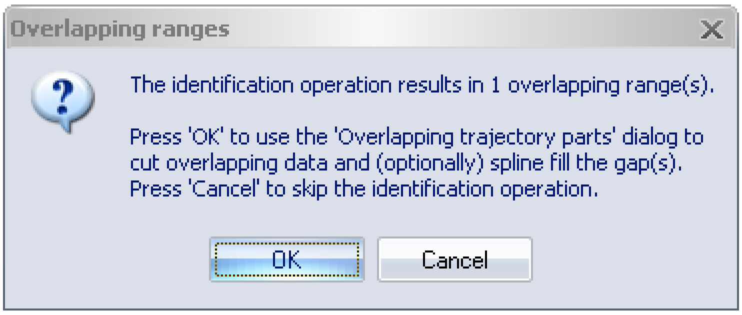
The trajectories can be moved to another **Trajectory info** window by a simple drag and drop, either one by one or in a group. If the trajectory is dropped in the **Labeled trajectories** window the trajectory is labeled 'New 0000', where the number depends on how many trajectories there are in the window. Use the options **Move label up** and **Move label down** in the **Trajectory info window** menu to change a position of a trajectory in the Labeled trajectories window

Trajectories can also be joined by drag and drop. Select a trajectory and drag and drop it on another trajectory or on an empty label. The trajectory will then contain a new part. If the trajectories overlap there will be a clash error, see chapter “Overlapping trajectory parts” below. Trajectories where one of the trajectories or a part is completely overlapped by the data of the other trajectory cannot be joined at all.

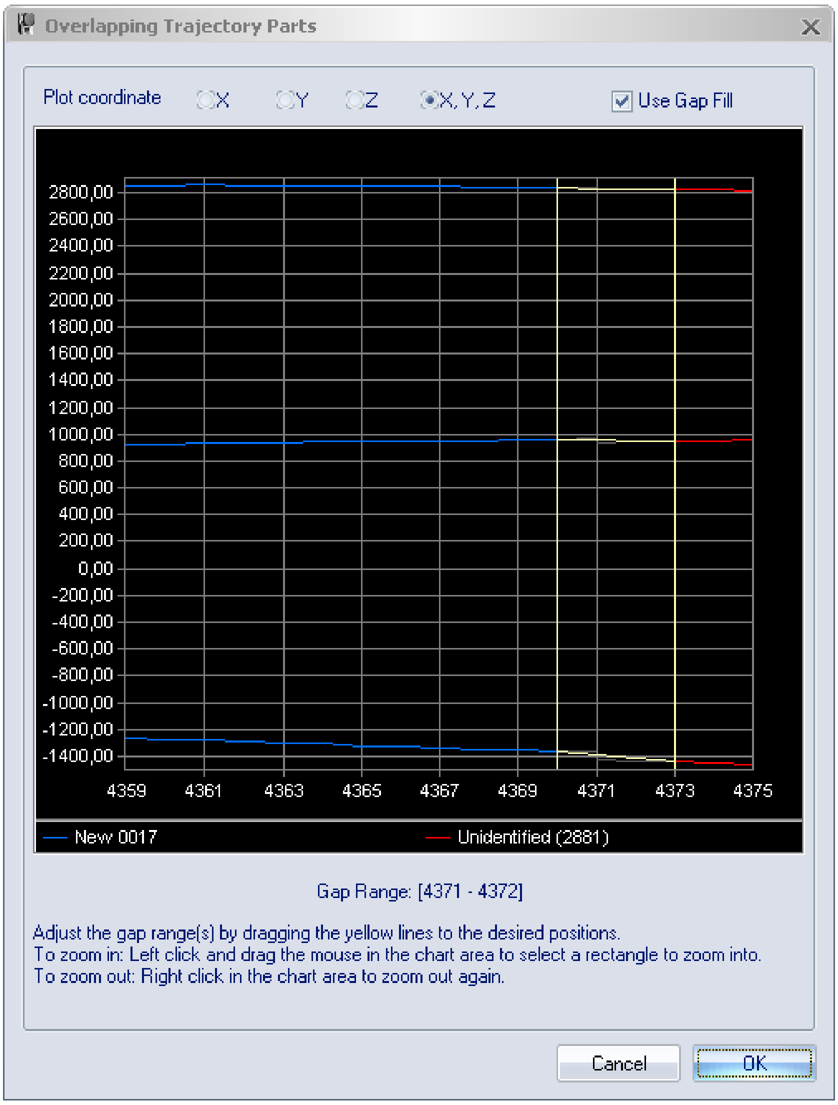
Note: The trajectories can also be drag and dropped from the 3D view window, see chapter “Trajectories in 3D views” on page 54.

Overlapping trajectory parts

Two trajectories cannot be joined as long as they have data in the same frames. This will result in a clash and then the **Overlapping ranges** dialog is displayed.



Click **OK** to open to the **Overlapping trajectory parts** dialog, with which the correct data can be chosen for the joined trajectory.



The data of the trajectories are shown in the chart area of the **Overlapping trajectory parts** dialog. This is used to visualize the data which will be included in the joined trajectory. The data is shown for the X, Y and Z coordinates, where the coordinate that is displayed is chosen with the **Plot coordinate** option. The names and colors of the trajectories are shown in the legend box below the plot area.

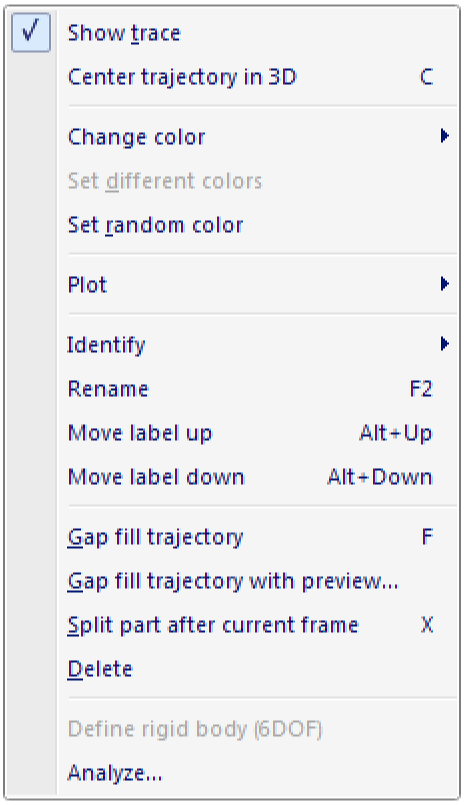
Move the range selectors (the yellow lines) to choose the data that will be included in the joined trajectory. Choose a range where you delete the data that seems to be wrong. It is sometimes better to delete some extra frames and gap fill that part instead of using the original data. The area between the range selectors can either be gap filled or deleted, which is determined by the **Use Gap fill** option. If the gap fill function is used the gap filled part is shown as a yellow line between the range selectors.

To zoom in on an area of the plot, click and drag a rectangle in the plot area. Zoom out to the previous zoom by right-clicking in the area.

If there is more than one clash, the **Next** and **Previous** options can be used to step between the gap ranges. The range and the number of the gap are shown below the plot.

Note: The area between the range selectors must always include at least two frames, which must be gap filled or deleted.

Trajectory info window menu



The **Trajectory info window** menu contains options for the tracked trajectories in a file. Most of the options are not available in real-time. The menu can be accessed by right-clicking in the following places:

- On a trajectory or a part in one of the **Trajectory info** windows.
- On a marker or a trace in a 3D view window.
- In the empty space of a 3D view window.
- It is also available on the **Edit** menu.

When multiple trajectories are selected the options on the menu is applied to all of the trajectories. The following options are available on the menu:

Show trace
Toggle the display of the trace of the selected trajectories.

Center trajectory in 3D
Center the 3D view on the selected trajectory or part. The time will be moved to the first frame of the selected marker, if the trajectory is not visible in the current frame.
Note: This option is available in RT/preview mode.

Change color
Change the color of the selected trajectories.

Set different colors
Change the color of the selected trajectories to different colors. It steps through 256 colors, but with different steps depending on the number of trajectories.

Set random color
Change the color of the selected trajectories to random colors, this can result in similar colors.

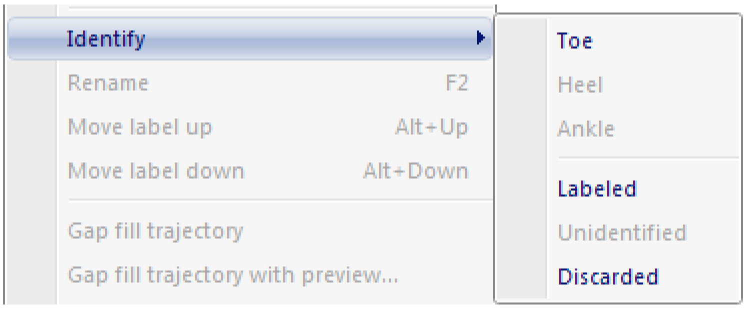
Plot

Plot the 3D data of the selected trajectories, see chapter “Plot 3D data” on the facing page.

Note: This option is available in RT/preview mode.

Identify

Move the selected trajectories to another **Trajectory info** window. It is also possible to move one trajectory to a label in the **Labeled trajectories** window.



Rename

Rename the selected trajectory. This is the same as double-clicking on the label.

Note: This option can only be used on one trajectory at a time.

Move label up

Move the label up one step in the Labeled trajectories list.

Move label down

Move the label down one step in the Labeled trajectories list.

Gap fill trajectory

Gap fill the selected trajectories using the NURBS interpolation, see chapter “Gap fill trajectories” on page 72.

Gap fill trajectory with preview…

Open the **Gap fill trajectory** dialog to see a preview of the gap filled part, see chapter “Gap fill trajectory with preview” on page 73 .

Split part after current frame

Split a trajectory or part of a trajectory after the current frame into two parts, see chapter “Split part after current frame” on the facing page.

Delete

Delete the selected trajectories or parts, see chapter “Delete trajectories” on the facing page.

Define rigid body (6DOF)

Create a new 6DOF body on the **6DOF bodies** page, the points of the body is defined by the selected trajectories’ average positions in all of the frames. The 6DOF body will be added both to the settings in Project options and in the current file.

Note: Make sure that the body includes at least three points.

Note: The trajectories will keep their label name unless they are named 'New XXXX' or are unidentified.

Analyze…

Calculate parameters from the 3D data and/or filter the 3D data, see chapter “Analyze” on page 74. Analysis can also be accessed from the **Analyze trajectory** button on the **AIM** toolbar.

Plot 3D data

To plot 3D data select one or more trajectories, click **Plot** on the **Trajectory info window** menu and then the type of data. The data that can be plotted are: **3D X-position**, **3D Y-position**, **3D Z-position** and **Residual**. For information about the **Plot** window see chapter “Plot window” on page 85.

The curve of each trajectory will have the same color as in the **Trajectory info** windows. Trajectories in the **Unidentified trajectories** window can be separated by their sequential number.

If trajectories with the same or similar colors are plotted the plot will automatically use different colors. This means that if you used **Set different colors** on a large number of trajectories then the color of two trajectories close to each other in the list may change their color in the plot.

Split part after current frame

Split part after current frame (shortcut 'x') on the **Trajectory info window** menu splits a trajectory or a part of trajectory into new parts after the current frame. The first part will end at the current frame and the other part will start on the frame after the current frame. This means that there is no gap between the two parts, but in the trace in the 3D view it is visualized as a gap between the two parts to show the split.

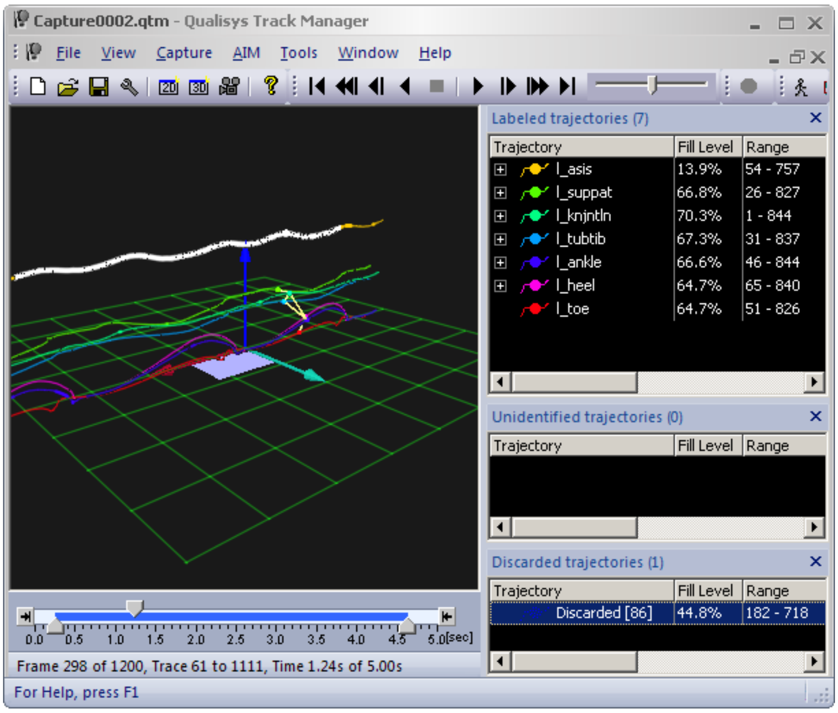
You can also use the **Cut trajectory trace** tool in the 3D view window to split a trajectory. Click on a trace with the tool to split the trajectory at that position.

Delete trajectories

To delete a trajectory or a part of a trajectory select it in a **Trajectory info** window and click **Delete** on the **Trajectory info window** menu. You can also use Delete on the keyboard, this will also work in a **3D view** window. When a trajectory is deleted from the **Labeled trajectories** or **Unidentified trajectories** window, the trajectory is moved to the **Discarded trajectories** window.

When **Delete** is used on a trajectory in the **Discarded trajectories** window, the trajectory will be deleted from the file. A confirmation dialog is displayed before the action is performed. The trajectory can, however, be retrieved by reprocessing the file.

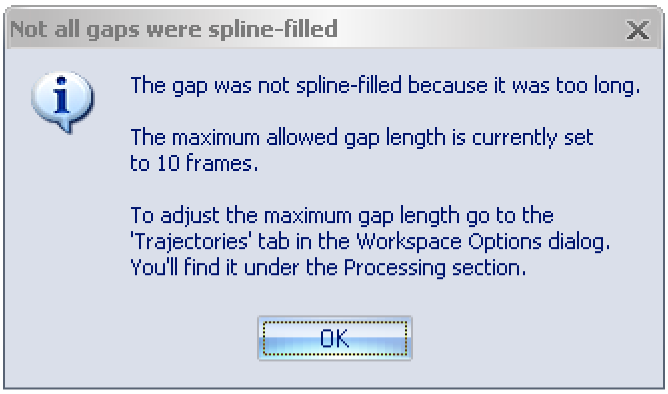
The trajectories in the **Discarded trajectories** window are not shown in the **3D view** windows and are not included in exported data. To show a discarded trajectory and its trace in the 3D view, select the trajectory in the **Discarded trajectories** window, see picture below.



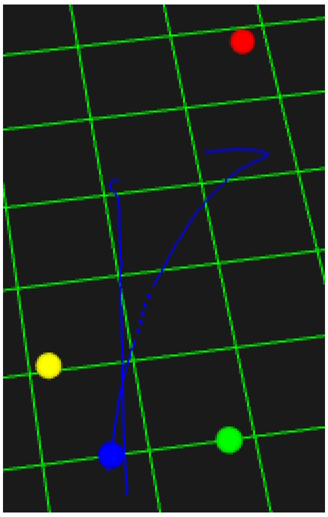
Gap fill trajectories

Gap fill is a function which calculates a probable path for the trajectory in the frame gap between two parts. Gap filling can be performed either as a processing step after a capture or manually to the selected trajectories in the file. You can also use the virtual markers from 6DOF data if you have markers that have rigid relation to each other, see chapter “How to use virtual markers in an AIM model” on page 297.

When gap fill is performed as a processing step it is tried on every trajectory. While when it is done manually it is only the selected trajectories that are gap filled. Whether a trajectory can be gap filled or not is determined by **Max frame gap**, on the **Trajectories** page in the **Project options** dialog. It specifies the number of empty frames that is allowed between two parts to gap fill the gap, if the gap is too large the following dialog appears.



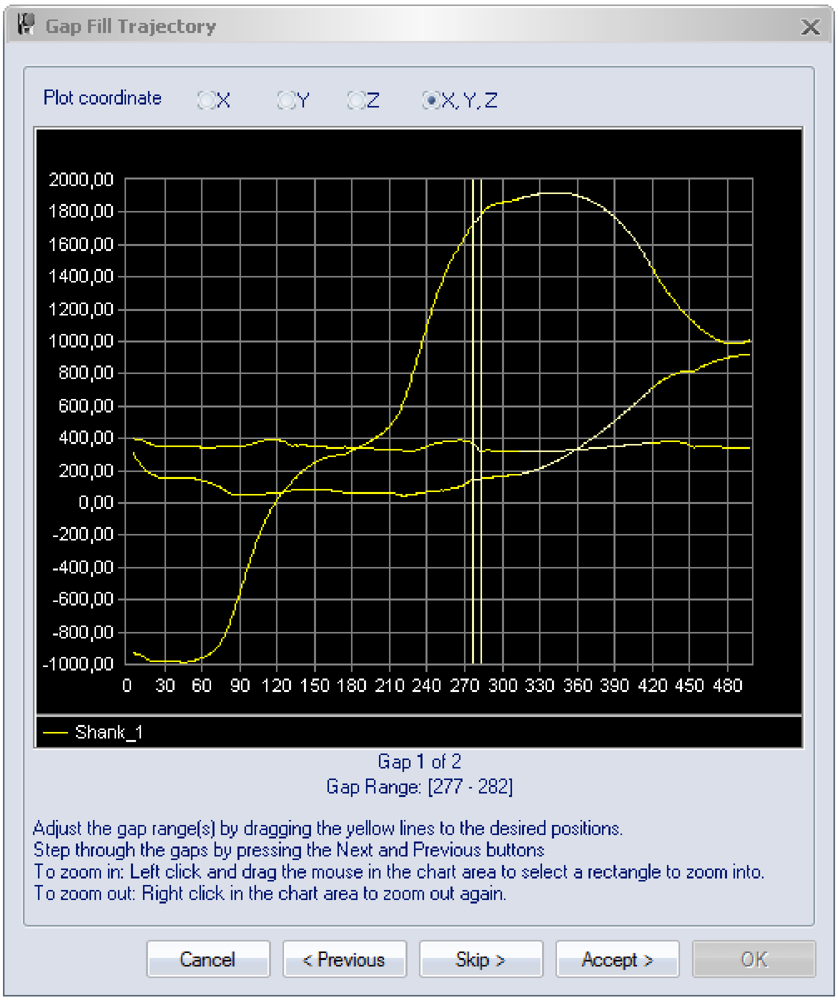
To gap fill in an active capture file, right-click on the selected trajectories in the **Trajectory info** window and click on **Gap fill trajectory** in the **Trajectory info window** menu. A gap filled part, which is represented by a dotted line in the 3D view, is added to the trajectory.



Gap fill trajectory with preview

The **Gap fill trajectory with preview** option opens the **Gap fill trajectory** dialog, which shows a preview of the gap filled parts and where the size of the gaps can be edited.

Note: If several trajectories are selected, only one trajectory at a time is opened in the **Gap fill trajectory** dialog. Click **OK** in the first dialog to go to the next trajectory.



The data of the trajectory is shown in the chart area of the **Gap fill trajectory** dialog. This is used to visualize the gap filled parts, which will be added to the trajectory. The data is shown for the X, Y and Z coordinates, where the coordinate that is displayed is chosen with the **Plot coordinate** option. The name of the trajectory is shown in the legend box below the plot area.

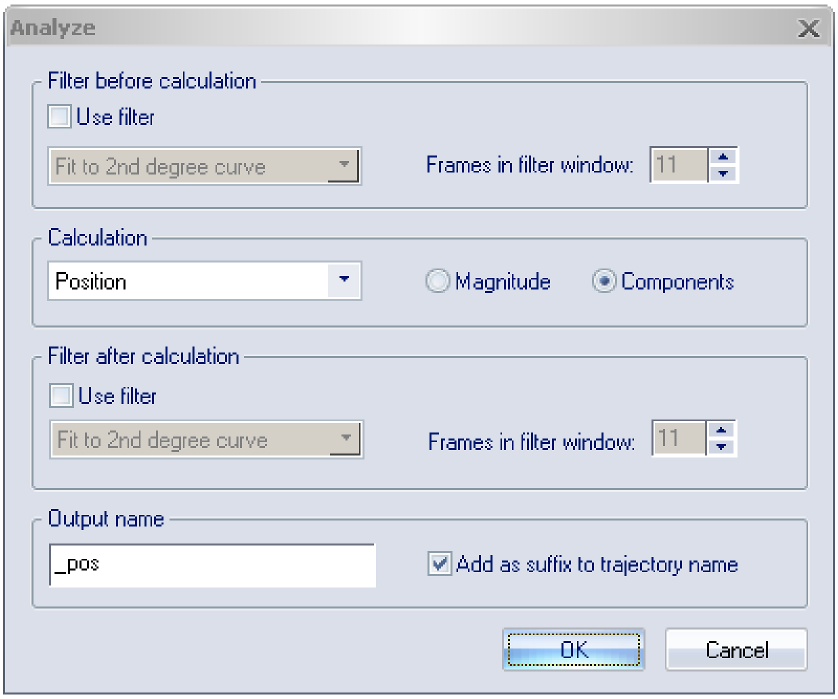
Move the range selectors (the yellow lines) to change the size of the gap. The gap cannot be smaller than the actual gap, but it can be made larger. If it is enlarged the earlier data inside the yellow lines are deleted and replaced with the gap filled part. It is sometimes a good idea to delete some of the data before and after the gap because those parts may be noisy.

To zoom in on an area of the plot, click and drag a rectangle in the plot area. Zoom out to the previous zoom by right-clicking in the area.

If there is more than one gap, the **Accept**, **Skip** and **Previous** options can be used to step between the gaps. The **Accept** option will apply the suggested gap fill and step to the next gap. The **Skip** option will just step to the next gap. Step back with the **Previous** option, if you want to change a previous gap fill. The range and the number of the current gap are shown below the plot.

Analyze

Calculations and filters can be applied to the trajectory data with **Analyze** on the **Trajectory info window** menu, . Select the trajectories that you want to analyze and click on **Analyze** to open the **Analyze** dialog. The dialog can also be opened with the **Analyze trajectory** button on the **AIM** toolbar.



When you click **OK** in the **Analyze** dialog a new dialog appears, see chapter “Analysis Calculation” on page 77.

Filters

Filters can be applied both before and after the calculation, by selecting **Use filter** under the respective heading. Set the number of frames that are used for each point in the filter with the **Frames in filter window** option. The number of frames must have an odd value.

There are two available filters:

Fit to 2nd degree curve
This filter uses a 2nd degree curve when processing the data. For each frame, the filter will first find the 2nd degree curve that best fits the data in the filter window around the current frame. Then the data of the current frame is set to the value of that curve at the current frame.

Moving average
For each frame, this filter first finds the average of the data in the filter window around the current frame. Then the filter sets the data of the current frame to the average found. (This can also be seen as fitting the data of the filter window to a polynomial of degree zero.)

Calculation

Under the **Calculation** heading there are five parameters that can be calculated:

Position
No calculation is performed. Use this setting to filter the motion data, together with selecting **Use filter** either before or after calculation. Unless the data is filtered or you select the **Magnitude** radio button there is no difference between the result of the analysis and the original data.

Velocity
Calculates the velocity of the trajectories. Select **Magnitude** to calculate the speed.

Acceleration
Calculates the acceleration of the trajectories.

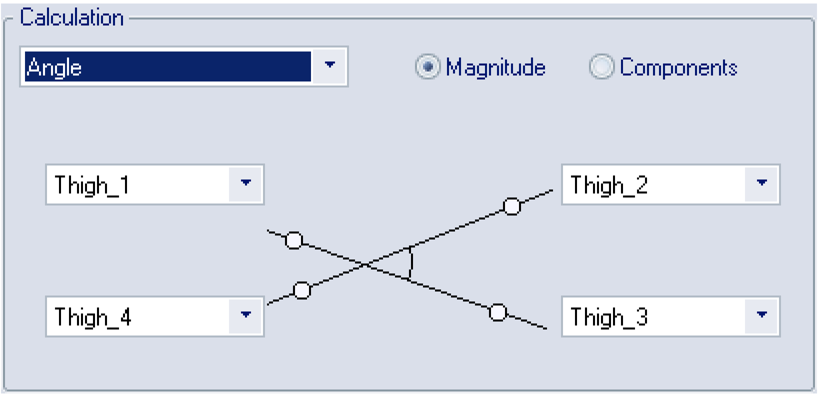
Distance
Calculates the distance between two trajectories over time.

Note: Distance can only be calculated when just two trajectories are selected.

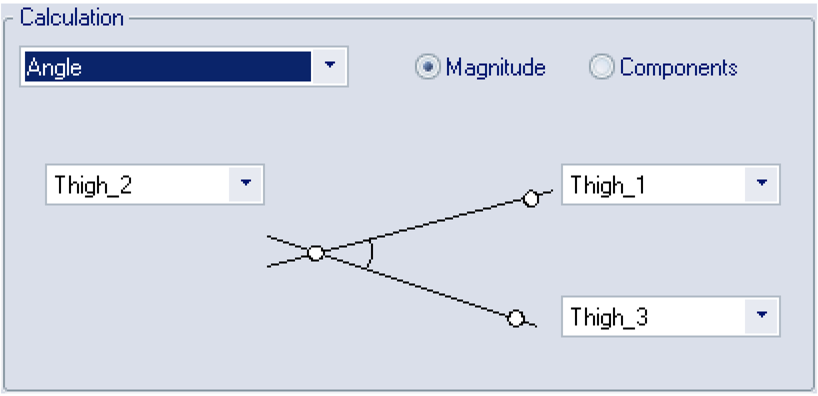
Distance traveled
Calculate the distance that the trajectory has traveled in the measurement. The distance will increase with every frame that marker position has changed more than 0.2 mm compared with the last time the marker moved. The 0.2 mm hysteresis is there to remove the static noise, otherwise a marker that is static will still get a distance traveled over time. If there is a gap in the marker data the distance traveled will restart on 0.

Important: Because noise will be added in this sort of analysis the accuracy can be low depending on the settings. The error will be larger with a high capture rate, because for every sample more noise is added to the distance. And even with the hysteresis it is recommended to use a filter to remove noise. Also remove any data in the trajectory that you do not want to add to the measurement, because spikes in the static data will not be removed by the filter or the hysteresis.

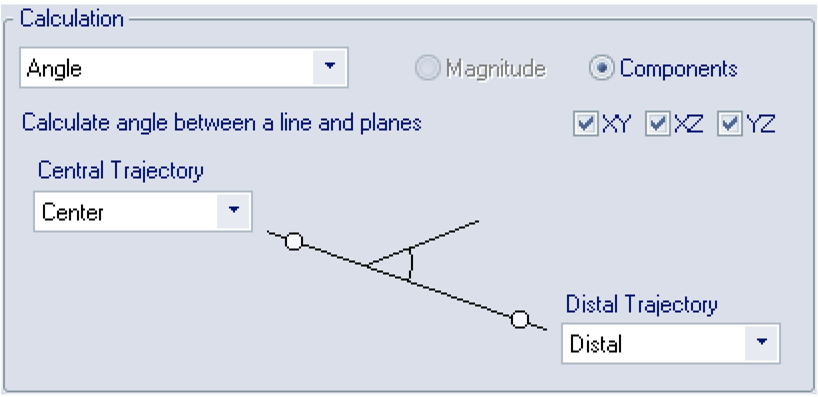
Angle
Calculates the angle between two lines in space, using either two, three or four trajectories. Select the trajectories that you want to match with the markers in the diagram to calculate the angle between the lines. For three and four markers the angles are calculated in the range 0 – 180 degrees. The look of the dialog when four trajectories are selected is shown below.



When three trajectories have been selected the dialog looks like this:



When two trajectories have been selected the dialog looks like this:



For two markers the angle is calculated between the line and the planes of the coordinate system. Select which planes you want to use with the checkboxes **XY**, **XZ** and **YZ**. The angle is calculated between -90 and 90 degrees. Which sign the angle has depends on how line from **Center Trajectory** to **Distal Trajectory** is pointing. For example if you are using the XY plane then the positive side is when the line from central is pointing in the positive Z direction.
The angle between the line and an coordinate axis is the complementary angle to the perpendicular plane according to this formula, "angle to axis"="angle to perpendicular plane" - 90 degrees.

Angular velocity
Calculates angular velocity for an angle defined in the same way as for the angle calculations (see above). The angular velocity is the first derivative of an angle, i.e. the rate of change of an angle, in degrees per second.

Note: Two to four trajectories must be selected to calculate angular velocity.

Angular acceleration
Calculates angular acceleration for an angle defined in the same way as for the angle calculations (see above). The angular velocity is the second derivative of an angle, i.e. the rate of change of an angular velocity, in degrees per second.
Note: Two to four trajectories must be selected to calculate angular acceleration.

For each of the calculations it is possible to choose whether the output will be the **Magnitude** or the **Components** of the results.

- For **Position**, **Velocity**, **Acceleration** and **Distance**, the **Components** means that the result is shown for the three coordinates (X, Y and Z) separately. The components of **Distance** are the distance projected on the three axes (X, Y and Z).
  The **Magnitude** means that the result is shown as the length of the vector made up of the three components. For **Position** this means the distance in mm from the origin of the coordinate system and for **Velocity** this means the speed (in mm/s) of the trajectories. For **Acceleration** it does not have a separate name, it is simply the magnitude of the acceleration. For **Distance**, the magnitude is probably the result that is most appropriate, since it is the actual distance between the two trajectories.
- For **Angle** and **Angular velocity**, the **Components** means that the angle is projected onto the three perpendicular planes (YZ, XZ and XY respectively), while the **Magnitude** is simply the angle between the two arms.

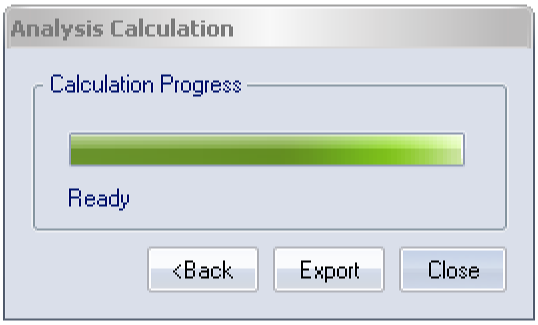
Output name

The name under the **Output name** heading is used in the **Plot** window and in the TSV file. When the **Add as suffix to the trajectory name** setting is selected the trajectory name is added before the **Output name**.

The **Add as suffix to the trajectory name** setting can only be changed when a single trajectory is selected. If more than one trajectory is selected and **Position**, **Velocity** or **Acceleration** calculations are performed, the setting is always selected. Otherwise all the results would have the same name. On the other hand, when **Angle**, **Angular Velocity** or **Distance** calculation is performed, the setting is always deselected, since there are two, three or four trajectories selected but the result is a single value.

Analysis Calculation

When you click **OK** in the **Analyze** dialog, the following dialog appears:



The dialog reports the progress of the calculations. If you select many trajectories and perform an acceleration calculation with filtering both before and after, the calculations may take some time.

When the calculation is finished, the text "**Ready**" is displayed below the progress bar and the result is shown in a **Plot** window. The results can also be exported to TSV format with the **Export** option. Click **Back** to go back to the Analyze dialog and perform another calculation using the same trajectories. Click **Close** to close the dialog.

Label lists

The labels of the trajectories in the **Labeled trajectories** window can be saved in label lists. The label lists are controlled from the **Labels** menu.

To save a label list from the current capture file, click **Save** on the **Labels** menu. Enter the name of the list and click **Save**. The label list file is saved in the **Labels** folder in the project folder. The label list is saved in standard text files and contains the name and color of the labeled trajectories and bones between the markers in the 3D view.

A label list is loaded by clicking **Load** on the **Labels** menu. If labels have been added manually to the **Labeled trajectories** window they will be overwritten when a label list is loaded. The trajectories will therefore be renamed.

Labels menu

The **Labels** menu can be opened from the **Edit** menu or by either right-clicking in the empty area of the **Labeled trajectories** window or on the names of the columns.

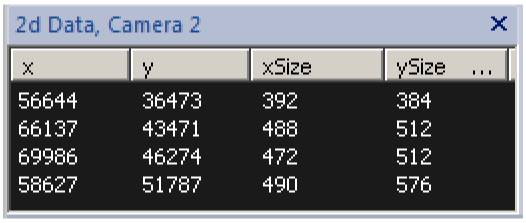
Add new label
Add a new empty label to the **Labeled trajectories** window.

Load label list...
Load a label list to the **Labeled trajectories** window.
Note: If labels have been added manually to the **Labeled trajectories** window they will be overwritten when a label list is loaded. The trajectories will, however, just be renamed.

Save label list...
Save the label list to a text file.

Default label list...
Go to the **Trajectories** page in the **Project options** dialog to specify a default label list..
Note: This label list is loaded automatically at the next measurement session.

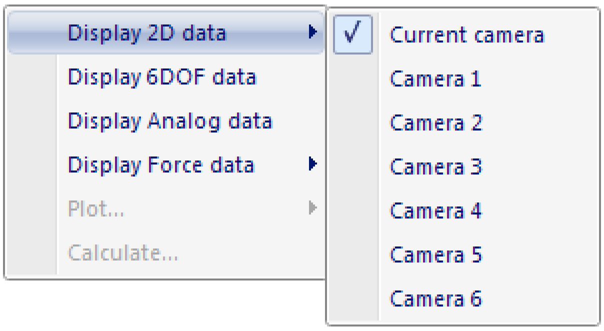
Data info window



The data for the current frame can be viewed in the **Data info** window. Open this window by clicking **Data info window [Ctrl+D]** on the **View** menu. It can be opened both in file mode and in preview mode. It is a tool window and is always placed on the left side of the main window.

All of the data which has been measured and processed with QTM can be viewed in the **Data info** window. For information about the different data types see chapter "Data types" on next page.

Data info window menu



The **Data info window** menu is accessed by right-clicking in the **Data info** window. With the **Data info window** menu the data can be switched between the data types and plotted. The following options are available on the menu:

Display 2D data

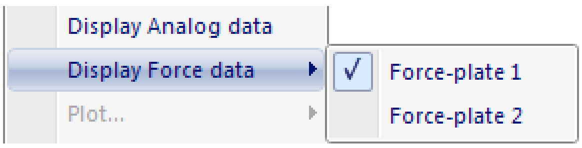
Current camera
Display the current camera, i.e. the last the camera that you clicked on, in the 2D view window.

Camera …
Choose which camera's 2D data that will be displayed.

Display 6DOF data
Display the 6DOF data of the motion capture.

Display Analog data
Display the analog data of the analog board.

Display Force data



Force-plate …
Choose which force-plate's force data that will be displayed.

Plot
Plot the parameters for the selected data, see chapter “Plot window” on page 85.

Calculate
Calculate magnitude of distance from origin of a 6DOF body, see chapter “Calculate” on page 84.

Data types

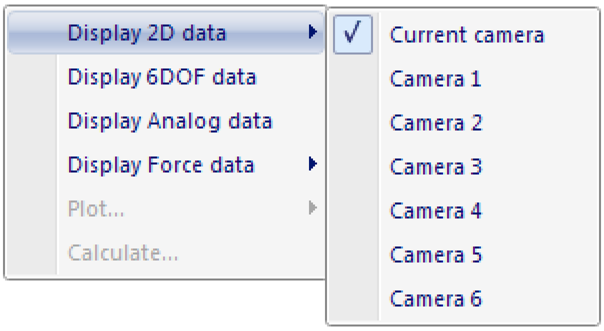
In the **Data info** window the four following data types can be shown: 2D, 6DOF, analog and force. The data types and how to plot them are described in the sections below.

6DOF data information

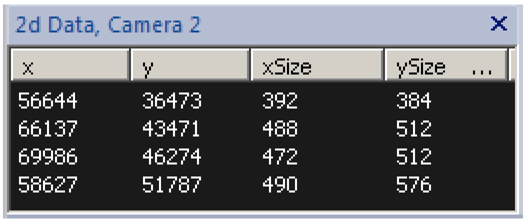
Analog data information

Force data information

2D data information



Click **Display 2D data** in the **Data info window** menu and then click a camera to show its 2D data in the **Data info** window. Data can only be shown for one camera at a time. Use **Current camera** to display the data for the current camera in the 2D view window.

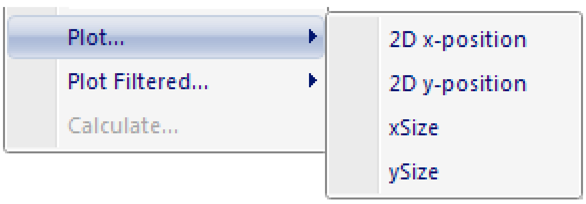

2d Data, Camera 2

x	y	xSize	ySize ...
56644	36473	392	384
66137	43471	488	512
69986	46274	472	512
58627	51787	490	576

When 2D data is displayed, each row in the **Data info** window represents a marker. The data of the markers is for the current frame and it is displayed in the following four columns:

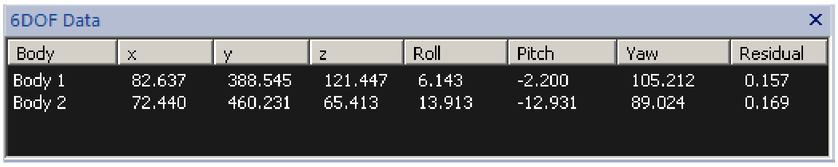
x, y
The position of the marker on the sensor in camera units. The 2D data in this window is unlinearized.

xSize, ySize
The size of the marker in per mill of the diagonal FOV of the camera.



To plot the data select the data for one or more markers, click **Plot** or **Plot filtered** on the **Data info window** menu and then the type of data. With **Plot filtered** you can apply a **Fit to 2nd degree curve** or **Moving average** filter. For information about the **Plot** window see chapter “Plot window” on page 85.

6DOF data information



6DOF Data

Body	x	y	z	Roll	Pitch	Yaw	Residual
Body 1	82.637	388.545	121.447	6.143	-2.200	105.212	0.157
Body 2	72.440	460.231	65.413	13.913	-12.931	89.024	0.169

The data of the 6DOF bodies in the current frame can be viewed in the **Data info** window. The bodies will be shown in the same order as on the **6DOF bodies** page and the data will use the definitions for angles and local coordinate system on the **Euler angles** page respectively the **6DOF bodies** page.

Click **Display 6DOF data** in the **Data info window** menu to show the 6DOF data in the following eight columns:

Body
The name of the 6DOF body.

x, y and z
The position (in mm) of the origin of the measured rigid body’s local coordinate system. The distance is to the origin of the coordinate system for rigid body data, see chapter “Coordinate system for rigid body data” on page 184.

Roll, Pitch and Yaw
The rotation (in degrees) of respectively the X-axis, Y-axis and Z-axis in the local coordinate system compared with the orientation of the coordinate system of the motion capture.

Note: The rotation is positive when the axis rotates clockwise when looking in the positive direction of the axis and it is applied in the order: roll, pitch and yaw. E.g. to mimic the rotation of a certain rigid body in a certain frame follow these steps:

a. Align the local coordinate system with the global.

b. Rotate roll to the correct angle.

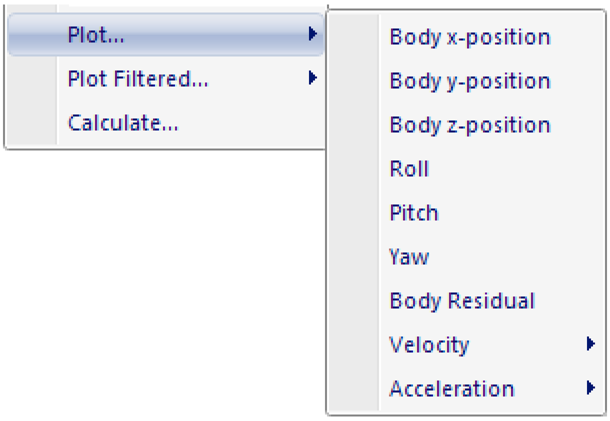
c. Rotate pitch to the correct angle. Remember that the Y-axis has moved because of the roll.

d. Rotate yaw to the correct angle. Remember that the Z-axis has moved because of roll and pitch.

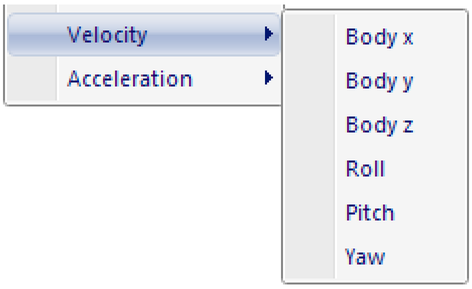
Note: If you change the Euler angle definition the example above will not be correct and the name of the angles can be changed.

Residual
The average of the errors (in mm) of each measured marker compared to the 6DOF body definition. This error is probably larger than the 3D residual.

To plot the data select the data for one or more 6DOF bodies, click **Plot** or **Plot filtered** on the **Data info window** menu and then the type of data. With **Plot filtered** you can apply a **Fit to 2nd degree curve** or **Moving average** filter.

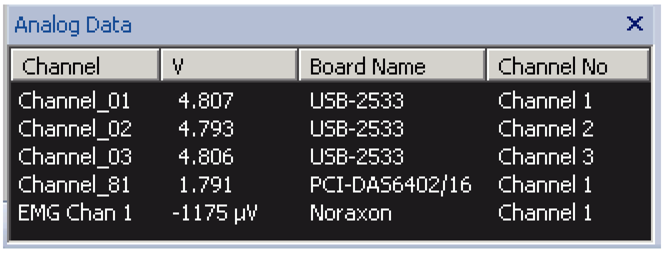


You can also plot the velocity and acceleration in the three directions of the coordinate system for rigid body data and the angular velocity and acceleration for the three rotation angles. It is recommended to use **Plot filtered** and apply the filter before the calculation for **Velocity** and **Acceleration**, because the noise is amplified by the calculations.



For information about the **Plot** window see chapter “Plot window” on page 85.

Analog data information



Channel	V	Board Name	Channel No
Channel_01	4.807	USB-2533	Channel 1
Channel_02	4.793	USB-2533	Channel 2
Channel_03	4.806	USB-2533	Channel 3
Channel_81	1.791	PCI-DAS6402/16	Channel 1
EMG Chan 1	-1175 µV	Noraxon	Channel 1

Click **Display Analog data** in the **Data info window** menu to show the analog data of the analog board in the **Data info** window. It will then show the data for the current frame in the following two columns:

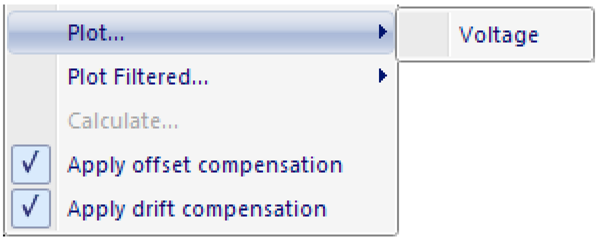
Channel
The name of the analog channel.
Note: The names can be changed under the **Analog boards** branch in the **Project options** dialog.

V
The voltage input to the analog board.

Board Name
The name of the analog board or of other analog device.

Channel No
The channel number on the analog device.

Right click on a analog channel to open the **Data info window** menu with the following settings.



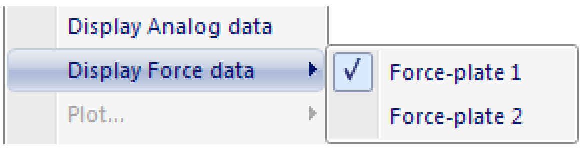
To plot the voltage, select one or more channels, click **Plot** or **Plot filtered** on the **Data info window** menu. With **Plot filtered** you can apply a **Fit to 2nd degree curve** or **Moving average** filter. For information about the **Plot** window see chapter “Plot window” on page 85.

Note: The number of seconds in the analog RT plot is specified on the **GUI** page in the **Project options** dialog, see chapter “GUI” on page 218.

Apply offset compensation
Activate/deactivate this option to apply/remove offset compensation for the selected channels, i.e. where the line is blue. The option can only be changed in a file and the settings used will be those set before the measurement on the **Analog board** page in the **Project options** dialog, see chapter “Compensate for analog offset and drift” on page 163.
In RT/preview however it will be greyed out and display the status of the compensation on the **Analog board** page. I.e. if the compensation is turned on the option is activated to display that it is applied in RT/preview.

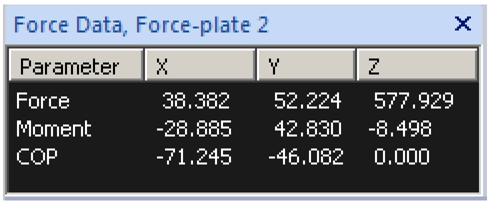
Apply drift compensation
Activate/deactivate this option to apply/remove drift compensation for the selected channels, i.e. where the line is blue. The option can only be changed in a file and settings used will be those set before the measurement on the **Analog board** page in the **Project options** dialog, see chapter“Compensate for analog offset and drift” on page 163.
In RT/preview the option is greyed out and deactivated since drift compensation is never applied in that mode.

Force data information



Click **Display Force data** in the **Data info window** menu and then the force plate to show the force data of a force plate in the **Data info** window. Data can only be shown for one force plate at a time.

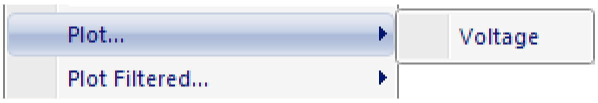
Note: To show the force data the calibration settings of the force plate must have been set on the **Force plate** page in the **Project options** dialog and the **Calculate force data** option must have been used during processing or **Recalculate forces** applied on the capture file, see chapter “Calculating force data” on page 311.



The **Data info** window will then contain data for the force plate's **Force**, **Moment** and **COP** (Center Of Pressure) vectors. The data is for the reaction force on the force-plate. It will show the data for the current frame in the following four columns:

Parameter
The name of the vector.

X, **Y**, **Z**
The size (in N, Nm respectively mm) of the vector in the internal coordinate system of the force plate. The Z direction of the force plate coordinate system is pointing down.

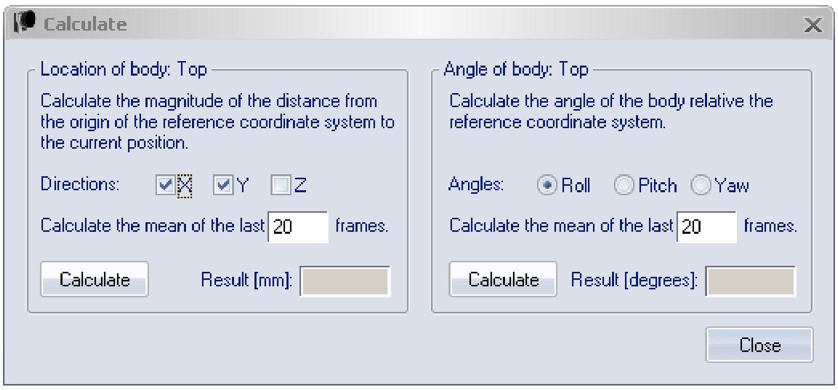


Note: The **Force** and the **COP** is also visualized with a vector in the **3D view** window. The vector is the opposite force applied to the force plate, i.e. opposite to that shown in the **Data info** window.

To plot a vector select one parameter, click **Plot** on the **Data info window** menu and then click **Parameter**. The data of the X, Y and Z directions are plotted for the selected parameter in one window. It is not possible to plot more than one parameter in one window. For information about the **Plot** window see chapter “Plot window” on the facing page.

The number of seconds in the force plot is specified on the **GUI** page in the **Project options** dialog, see chapter “GUI” on page 218.

Calculate



The **Calculate** dialog is used to calculate the magnitude of the distance from the origin of the reference coordinate system to the current position, under **Location of body** heading. Or the current rotation of the body under the **Angle of body** heading. Click on **Calculate** under the respective heading to get the result, the value will only be updated when you click **Calculate.**

The distance can be calculated for different planes, select the plane with the **X**, **Y** and **Z** options. The angle can only be calculated in one of the three rotations at a time. The data is calculate as the average of the data in the number of frames specified in the **Calculate the mean of the last** option.

Plot window

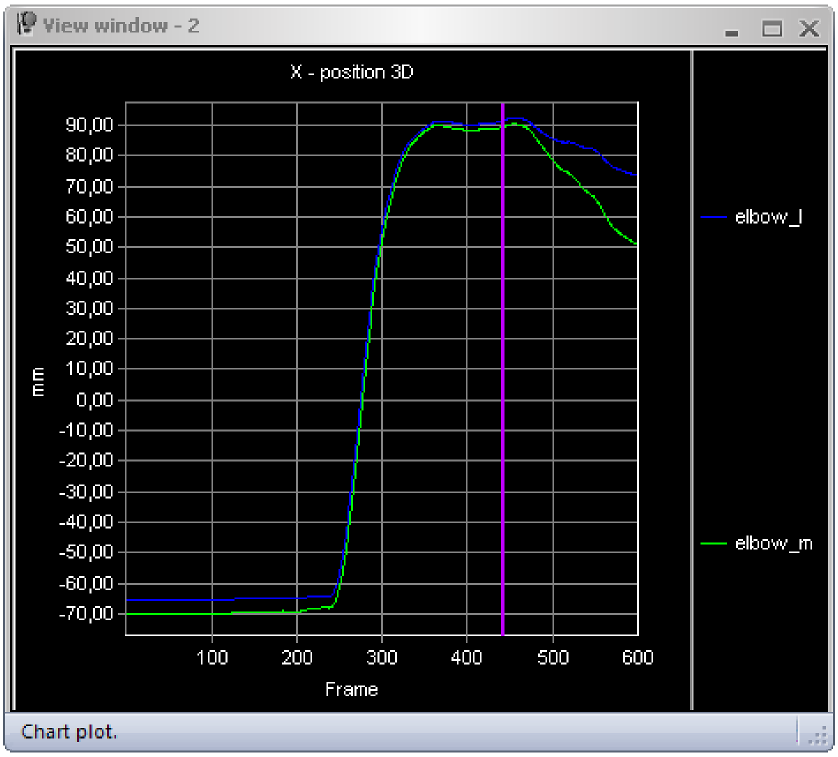
The **Plot** window displays a graph of the data that has been selected with the **Plot** command. It can be opened both in file and preview mode. In preview the number of frames in the plot depends on the width of the **Plot** window. The default window shows the graph and the **Legend** box, see screen dump below.

The **Legend** box displays the names and colors of the series. The series have the same name as the rows in the **Data info** window. If the graph contains 3D data of the trajectories, the color of the series are the same as the color of the trajectory.

Note: If the measurement range is smaller than the initial range, the parts of the trajectory which are outside the measurement range are grey.

The graph contains the following default graphics:

- A title above the plot, which shows what is plotted in the graph.
- A purple line in the plot, which represents the current frame in the **View** windows.
- An X-axis, which is always in frames.
- A Y-axis, where the unit depends on what is plotted in the window.



The settings for the **Plot** window are reached by right-clicking in the plot area, which will open the following menu.

The following options are available on the menu:

Toolbar
Toggle the display of the toolbar at the top of the **Plot** window, see chapter "Toolbar" below.

Data editor
Toggle the display of the **Data** editor, where the data of the series are shown.

Legend box
Toggle the display of the **Legend** box which contains the names and colors of the series, see chapter "Legend box" on page 88.

Gallery
Change the chart type.

Color
Change the color of the selected area.

Edit title
Edit the title of the chart.

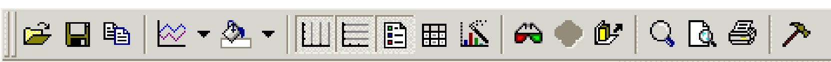
Point labels
Toggle the display of point labels, i.e. the data for each frame.

Font
Change the font of the point labels.

Properties
Open the **Properties** dialog. For more information about the settings in the **Properties** dialog click **Help** in the dialog.

Note: It is also possible to right-click on other items in the **Plot** window such as an axis or a series. The menu will then be limited to settings for that item.

Toolbar



The toolbar in a **Plot** window is opened with **Toolbar** on the menu of the **Plot** window. It contains the following options, from left to right:

Open chart
Open a saved chart or a chart template (Chart FX files or templates).

Save chart
Save a chart to one of the following formats:

Chart FX file
Save the data and the settings.

Chart FX template
Save only the settings.

Text file
Save the data of the chart as a text file with tab-separated values.

Metafile picture
Save as EMF file.

Bitmap picture
Save as BMP file.

Copy to clipboard
Copy the file to the clipboard with one of the following options:

As a bitmap

As a metafile

As text (only data)

To Office Application

Gallery
Change the chart type.

Color
Choose a color and drag and drop it to change the color of that area.

Vertical grid
Toggle the display of the vertical grid.

Horizontal grid
Toggle the display of the horizontal grid.

Legend box
Toggle the display of the **Legend** box which contains the names and colors of the series.

Data editor
Toggle the display of the **Data** editor, where the data of the series are shown.

Properties
Open the **Properties** dialog. For more information about the settings in the **Properties** dialog click **Help** in the dialog.

3D/2D
Toggle between 2D and 3D graphs.

Rotate
Open the **Rotate** page in the **Properties** dialog.
Note: Only available when the graph is plotted in 3D graphs.

Z-clustered
Toggle whether series are plotted on different rows in 3D graphs.

Zoom
Toggle the zoom. Make rectangles with the mouse in the plot area to zoom.

Print preview
Open the print preview.

Print
Print the graph.

Tools
Toggle the activation of the following tools:

Series Legend
Same as **Legend** box.

Data editor

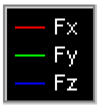
Toolbar

Palette bar
Toggle the display of the palette bar. The colors on the bar can be dragged and dropped to an area in the chart.

Pattern bar
Toggle the display of the pattern bar. The pattern on the bar can be dragged and dropped to chart elements.

Legend box

In the **Legend** box the names and colors of the series in the **Plot** window are listed, e.g. if the force data is plotted the three direction and their colors are displayed. The **Legend** box can be dragged and dropped inside the **Plot** window.



When right-clicking in the **Legend** box the following menu is opened.

The options on the menu are as follows:

Hide
Hide the **Legend** box.

Autosize
Automatically resize the **Legend** box.

Font
Change the font of the legends.

Fixed
Fix the **Legend** box in the **Plot** window.

Floating
Float the **Legend** box in a separate window.

Left, **Top**, **Right**, **Bottom**
Put the **Legend** box at an edge of the **Plot** window.

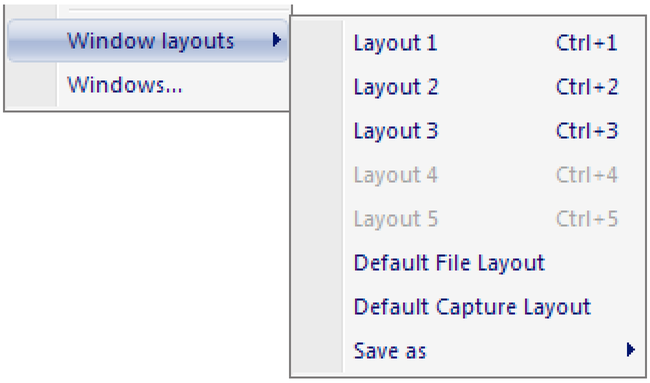
The window handling is done either manually with the mouse or with the commands in the **Window** menu. Manually you can move and resize the window to the desired place and size. While the following actions can be performed from the **Windows** menu:

- Open a new **2D**, **3D** or **Video view** window.
- Automatically change the placement of the **View** windows. The windows can be cascaded, tiled horizontally or tiled vertically.
- Activate and save window layouts, see chapter "Window layout" below.
- Edit the layout of the **View** windows, see chapter "Windows dialog" on next page.

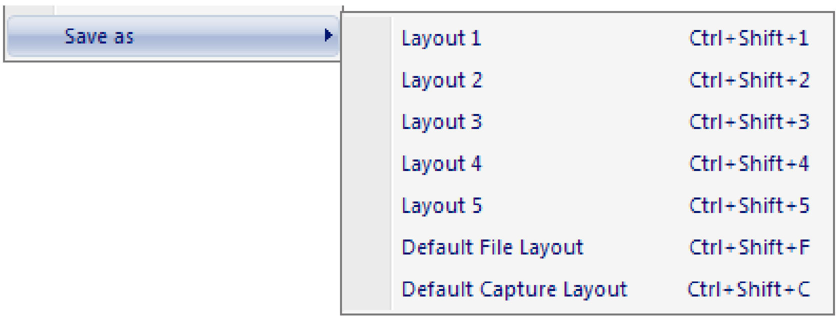
Window layout

With the window layouts you can save customized layouts, which include the placements of all of the windows in QTM. I.e. the layout is not limited to the **View** windows. The layouts are not saved with the capture file and can therefore be reused on any capture file.

To use window layouts click **Window layouts** on the **Window** menu. There are 5 shortcut layouts, which can also be applied with keyboard shortcuts (Ctrl + 1-5), and two default layouts. The default layouts are for file and capture mode and they are used when opening a saved file respectively when opening a new capture file before a measurement.



To save the current layout click **Save as** and then the desired layout. The 5 shortcut layouts can also be saved with the keyboard shortcuts Ctrl + Shift + 1-5.



The list below shows which objects that are saved in a window layout. For all of the objects the placement and size is saved. However, for some of the windows other properties are also saved in a layout, those properties are listed below the respective window.

2D view windows

The displayed cameras and their zoom.

3D view windows

Zoom and orientation of the coordinate system and the trace range.

Plot windows

The analysis or data plot which was used when saving the layout. However the file must have labeled trajectories with the same name as in the saved layout or the same data in the **Data info** window.

Note: If all labeled trajectories were selected for the plot, the labels of the trajectories are insignificant. And therefore the layout will work for any file with labeled trajectories.

The color scheme of the **Plot** window.

Toolbar

Data info window

The data type

Note: If the selected data type is not available in the file the 2D data is shown instead.

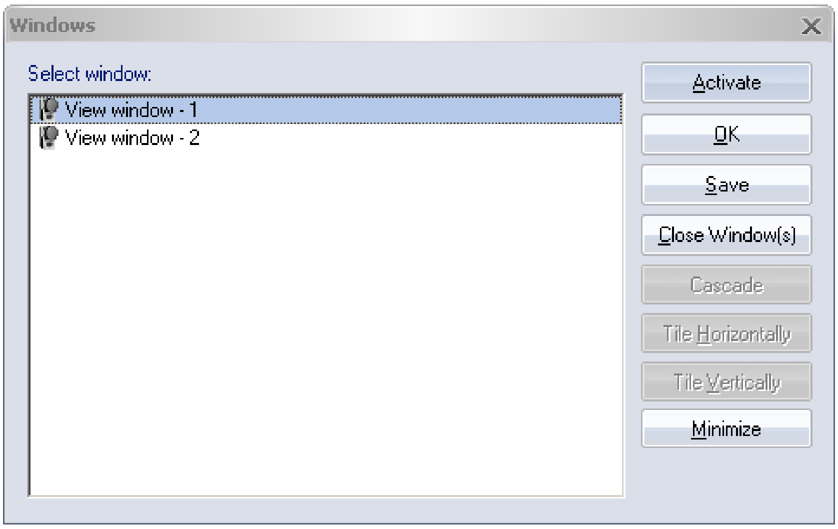
Trajectory info windows

Timeline

The display settings for the **Timeline**, not for example the measurement range.

Toolbars

Windows dialog



In the **Windows** dialog you can control which windows that will be shown. The following options are available in the dialog.

Activate
Move the selected window to the front and close the dialog.

OK
Close the dialog.

Save
Not implemented.

Close window(s)
Close the selected windows.

Cascade
Cascade the selected windows, the rest will be minimized.

Tile horizontally
Tile the selected windows horizontally, the rest will be minimized.

Tile vertically
Tile the selected windows vertically, the rest will be minimized.

Minimize
Minimize the selected windows.

Menus

File

New

Open a new empty capture file, i.e. go to preview mode. The cameras will start in the same mode as last preview or measurement.

Note: This is needed to start a new measurement.

Open

Open an existing capture file.

Note: QTrac files and TSV files can be imported with this option.

Batch process

Open configuration dialog for the batch processing function, see chapter “Batch processing” on page 279.

Save

Save a new capture file or save an existing file that has been changed.

Save as

Save the active file with a new name.

Close

Close the active capture file.

Switch project

Open the Switch project dialog switch projects, see chapter “Switch project” on page 11.

New project

Create a new project, see chapter “Creating a new project” on page 10.

Open project folder

Open the project folder where for example the calibration files are saved, see chapter “Project folder” on page 12.

Settings backup and restore

Open a dialog to save and restore backups of the project settings, see chapter “Backup of project settings” on page 12.

Export

Export the capture file to one of the following formats, see chapter “Data export” on page 314.

To TSV

To DIFF

To C3D

To MAT

Directly into Matlab

Import

Add link to video file

Import a link to an AVI file, see chapter “Import video link” on page 366.

Tema 2D file
Import 2D data from a Tema txt-file, see chapter “Analyzing data in video files” on page 261.

Load old workspace file
Load a QWR file with settings from QTM 2.5 or earlier, see chapter “Loading an old workspace file” on page 277.

1-0
The last ten opened QTM files.

Exit
Exit the application.

Edit

Undo
Undo the last edit action on trajectories.

Redo
Redo the last edit action on trajectories.

Delete
Delete trajectories or **Delete bones** depending on what is selected in the file.

Trajectory
The **Trajectory info window** menu for the selected trajectories, see chapter “Trajectory info window menu” on page 69.

Bone
The **Bone** menu for creating and deleting bones, see chapter “Bone menu” on page 61.

Labels
The **Labels** menu for the labels in the **Labeled trajectories** window, see chapter “Labels menu” on page 78.

Events
The **Events** menu for creating and editing events, see chapter “Events menu” on page 63.

View

Trajectory info windows
For information about **Trajectory info** windows see chapter ”Trajectory info windows”.

Labeled
Toggle the display of the **Labeled trajectories** window, which shows the trajectories that have been identified and labeled.

Unidentified
Toggle the display of the **Unidentified trajectories** window, which shows the trajectories that are unidentified.

Discarded
Toggle the display of the **Discarded trajectories** window, which shows all trajectories that have been discarded.

All
Toggle the display of all three **Trajectory info** windows: **Labeled trajectories**, **Unidentified trajectories** and **Discarded trajectories**.

Toolbars
For information about toolbars see chapter ”Toolbars”.

Capture toolbar
Toggle the display of the **Capture** toolbar.

Playback toolbar
Toggle the display of the **Playback** toolbar.

AIM toolbar
Toggle the display of the **AIM** toolbar.

Standard toolbar
Toggle the display of the **Standard** toolbar.

Customize
Not yet implemented.

Data info window
Toggle the display of the **Data info** window, which shows and plots 2D, 3D, 6DOF, analog and force data in both preview and file mode. For detailed information see chapter “Data info window” on page 79.

Timeline
For information about the **Timeline** and the menu see chapter “Timeline control bar” on page 61.

Status Bar
Toggle the display of the **Status** bar at the bottom of the main window.

File information
Show the start of the capture as time of the day and as a timestamp in seconds from when the computer was started. It also displays tracking residuals per camera see chapter “3D tracking test” on page 284.

3D view/2D view
3D view window menu or **2D view window** menu depending on the active View window, see chapter “3D view window menu” on page 60 respectively “2D view window menu” on page 44.

Play

Play/Stop
Switch between play and stop. The file will be played in the direction it was played the last time.

Play ▶

Stop ■

Play backwards ◀

Go to frame...
Enter a frame number in the **Timeline Parameters** dialog, see chapter “Timeline parameters” on page 63.

First frame |◀

Last frame ▶|

Next frame |▶

Previous frame

Forward five frames

Back five frames

Forward twenty frames

Back twenty frames

Forward a hundred frames

Back a hundred frames

Capture

Capture
Start the motion capture, see chapter “Capturing of measurements” on page 246. If used without preview the cameras will start in the same mode as last preview or measurement.
Note: Changes to **Stop capture** during a capture.

Calibrate
Open the **Calibration** dialog that is used to start the calibration, see chapter “Calibration dialog” on page 236.

Reprocess
Reprocess the active capture file, see chapter “Reprocessing a file” on page 281.

Recalculate forces F
Recalculate the force data in the active capture file, see chapter “Calculating force data” on page 311.

Linearize camera
Start linearization of a camera, see chapter “Linearize a camera” on page 229.

Run real time processing on file
Start real time process on the open file, see chapter “Real time processing on file” on page 272.

AIM

For information on automatic identification see chapter “Automatic Identification of Markers (AIM)” on page 302.

Apply model
Apply the current **AIM** model to the active capture file.
Note: The current **AIM** model is found on the **AIM** page in the **Project options** dialog.

Generate model
Generate an **AIM** model from the active capture file.

Generate from multiple files...
Generate an **AIM** model by selecting multiple files.

Tools

Project options
Open a dialog with settings for the QTM software, see chapter “Project options dialog” on page 104.

Customize
Not yet implemented.

Window

New window

2D
Open a new 2D view window.

3D
Open a new 3D view window.

Video
Open a new 2D view window displaying only the DV/webcam video cameras.

Cascade
Cascade the active **View** windows.

Tile horizontal
Tile the active **View** windows horizontally.

Tile vertical
Tile the active **View** windows vertically.

Window layouts
See chapter “Window layout” on page 89.

Layout 1, ... , **Layout 5**
Activate a saved window layout 1 to 5.

Default File Layout
Activate the default file layout.

Default Capture Layout
Activate the default capture layout.

Save as

Layout 1, ... , **Layout 5**
Save the current window layout as Layout 1 to 5.

Default File Layout
Save the current layout as default layout for file mode.
Note: The QTM software must be in file mode.

Default Capture Layout
Save the current layout as default layout for capture mode.
Note: The QTM software must be in capture mode (preview mode).

Windows
Open a dialog for the layout of the **View** and **Plot** windows. From the dialog a **View** window can be activated, i.e. brought to the front of the layout, see chapter “Windows dialog” on page 90. It also contains some of the options which are described above for the **Window** menu.

Help

Help topics
Open the QTM help window.

Display keyboard shortcuts
Show a dialog with all of the commands and their keyboard shortcuts.

Remote help instruction page at www.qualisys.com
Go to a website where you can start a remote help session to Qualisys with the Netviewer program.

Check for updates
If the computer has an internet connection, QTM will check if there is a newer version available on www.qualisys.com. If a newer version of QTM is available, information and a download screen will be presented.

About Qualisys Track Manager
View the user name, license number, version number, RT protocol and loaded modules of the installed QTM software. For information how to enter a plug-in license see chapter "Enter plug-in license" on page 7.

Popup window menus

All of the popup window menus are described under the relevant window, e.g. the **Trajectory info window** menu under the chapter about the **Trajectory info** window.

Toolbars

Standard toolbar

The **Standard** toolbar contains the following icons, from left to right:

New capture file

Open capture file

Save capture file

Open the **Project options** dialog

Undo

Redo

Open a new 2D view window

Open a new 3D view window

Open a new 2D view window displaying only the DV/webcam video cameras

Playback toolbar

The **Playback** toolbar contains the following icons, from left to right:

Go to the first frame

Go back twenty steps

Go back one step

Play reverse

Stop

Play forward

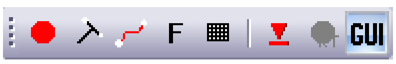
Go forward one step

Go forward twenty steps

Go to the last frame

Playback speed
Note: Move the **Playback speed** bar, which is logarithmic, or write the percentage of the speed directly in the **Playback speed** box.

Capture toolbar



The **Capture** toolbar contains the following icons, from left to right:

Start capture
Note: Changes to ■ (Stop capture) during a capture.

Calibrate the camera system

Reprocess the file

Recalculate the force data

Linearize camera

Add event

Save RT data (Not implemented yet)

Toggle GUI update
Note: The GUI is updated when the button is pressed down.

AIM toolbar

The **AIM** toolbar contains the following icons, from left to right:

Apply AIM model (in RT mode this button restarts AIM)

Generate AIM model

Analyze trajectory

Keyboard shortcuts

Menu shortcuts

The following menu commands can be accessed through keyboard shortcuts and are always available unless a dialog is open:

Ctrl + N
New capture file.

Ctrl + M
Open the **Start capture** dialog.

Ctrl + O
Open capture file.

Ctrl + S
Save capture file.

Ctrl + Z
Undo the last editing of the trajectories or bones.

Ctrl + Y
Redo the last action that was undone.

Ctrl + E
Add a manual event.

Ctrl + W
Open the **Project options** dialog.

Ctrl + D
Toggle the display of the **Data info** window.

Ctrl + 1, ... , Ctrl + 5
Activate a saved window layout.

Ctrl + Shift + 1, ... , Ctrl + Shift + 5
Save the current window layout.

F1
Open the QTM help window.

2D/3D view shortcuts

The following commands can be accessed in the 2D and 3D views:

2
Switch to 2D view.
Note: Only available in the 3D view.

3
Switch to 3D view.
Note: Only available in the 2D view.

C
Center on the selected marker in the **3D view** window.

L
Move the selected trajectories/parts to the **Labeled trajectory info** window.

U
Move the selected trajectories/parts to the **Unidentified trajectory info** window.

Del
Delete the selected trajectories/parts or selected bones.

F
Gap-fill the selected trajectories.

X
Split the selected trajectories after the current frame.

Ctrl + Alt
Quick identification method, see chapter “Manual identification of trajectories” on page 300.

Shift + C
Activate the **Center trajectory** tool.

Shift + X
Activate the **Cut trajectory trace** tool.

B
Create new bone between two or more selected markers.
Note: Only available in the 3D view.

Left-click
Select a trajectory.

Alt + Left-click
Select only the current part of the trajectory.

Shift + Left-click
Select multiple trajectories.

Ctrl + Left-click
Select multiple trajectories, possible to de-select trajectories.

Shift + Alt + Left-click
Select the current part of multiple trajectories.

Trajectory info window shortcuts

The following commands are available in the **Trajectory info** window:

C
Center on the selected marker in the **3D view** window.

L
Move the selected trajectories/parts to the **Labeled trajectory info** window.

U
Move the selected trajectories/parts to the **Unidentified trajectory info** window.

F
Gap-fill the selected trajectories.

X
Split the selected trajectories after the current frame.

Del
Move the selected trajectories to the **Discarded trajectories** window.

Ins
Add a new label in the **Labeled trajectories** window.

F2
Rename the selected trajectory.

Alt + Up
Move the trajectory up in the list.

Alt + Down
Move the trajectory down in the list.

Ctrl + A
Select all trajectories in the current **Trajectory info** window.

Up and Down arrows
Move in the list of trajectories. It can be used together with Shift to select trajectories.

Ctrl + Space
Toggle the selection of a trajectory.

Right and Left arrows
Expand respectively collapse a trajectory's list of parts.

Playback keys

The following keys can be used to play the file:

Space
Play forward and Stop.

Right Arrow
Go forward one step.

Shift + Right Arrow
Go forward five steps.

Ctrl + Right Arrow
Go forward twenty steps.

Ctrl + Shift + Right Arrow
Go forward one hundred steps.

Left Arrow
Go back one step.

Shift + Left Arrow
Go back five steps.

Ctrl + Left Arrow
Go back twenty steps.

Ctrl + Shift + Left Arrow
Go back one hundred steps.

Home
Go to the first frame.

End
Go to the last frame.

Project options shortcuts

The following keys can be used to navigate in the **Project options** dialog and in other similar dialogs:

Return
OK

Space
Click the selected button or toggle the selected option.

Up and Down arrows
Step through the alternatives in selected radio buttons.

Tab
Step through the settings. Use shift to step backwards.

Ctrl + Tab
Step out of the settings page to the options buttons at the bottom of the dialog. Use shift to step backwards.

Ctrl + Up or Down arrow
Step through the pages in the options list.

Ctrl + Right or Left arrow
Expand respectively collapse an options branch, e.g. **Processing**.

Dialogs

QTM dialogs

The dialogs in QTM where different settings are specified are described in the chapters were they are used. The essential dialogs are:

Project options dialog
This is the main dialog in QTM with the settings for the hardware and the processing of data, see chapter "Project options dialog" below.

Calibration dialog
This dialog is used to start a calibration, see chapter "Calibration dialog" on page 236.

Calibration results dialog
This dialog shows the results of a calibration, see chapter "Calibration results" on page 237.

Recalibration settings dialog
This dialog is used to recalibrate a capture file, see chapter "Reprocess calibration" on page 244.

Start capture dialog
This dialog is used to start a capture, see chapter "Start capture" on page 247.

File reprocessing dialog
This dialog is used to reprocess a capture file, see chapter "Reprocessing a file" on page 281.

Project options dialog

The configuration of software and hardware settings in QTM is accessed via **Project options** on the **Tools** menu. The settings are described in detail in the chapters: "Capture" on page 107, "Processing" on page 170 and "GUI" on page 218. You can get a summary of the current settings under the **Camera system settings** heading on the **Camera system** page.

There are four options at the bottom of the **Project options** dialog.

Click **OK** to save the settings and apply it to the current measurement. It will also close the dialog.

Click **Apply** to save the settings and apply it to the current measurement without closing the dialog.

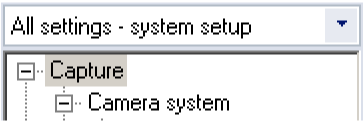
Click **Cancel** to close the dialog without saving the options.

Click **Help** to open the QTM help window.

The settings displayed in the **Project options** dialog can be changed with an option, see chapter "Change displayed pages in Project options" on the facing page.

Note: Most of the settings will only affect a measurement if QTM is still in the preview mode, i.e. before the measurement has started. To change settings on a file you must use the reprocessing dialog instead, see chapter "Reprocessing a file" on page 281.

Change displayed pages in Project options



Which pages that are displayed in the **Project options** dialog can be changed with the drop-down list in the top left corner. The three options are:

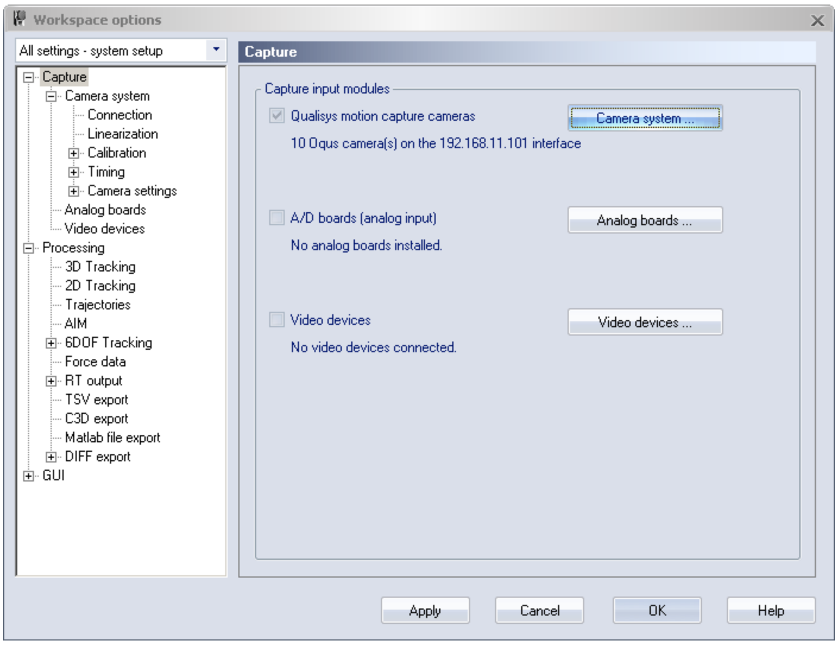
All settings - system setup
All of the pages are displayed in the settings tree. Use this option if you want to see all of the available settings.

Marine measurement settings
Only the pages that are regularly used during marine measurements are displayed in the settings tree.

Gait measurement settings
Only the pages that are regularly used during gait measurements are displayed in the settings tree.

Project options

Capture



The **Capture** page displays the current system setup of Cameras, Analog boards and Video devices. Click on **Camera System**, **Analog boards** or **Video devices** to go the settings for that input.

Camera system

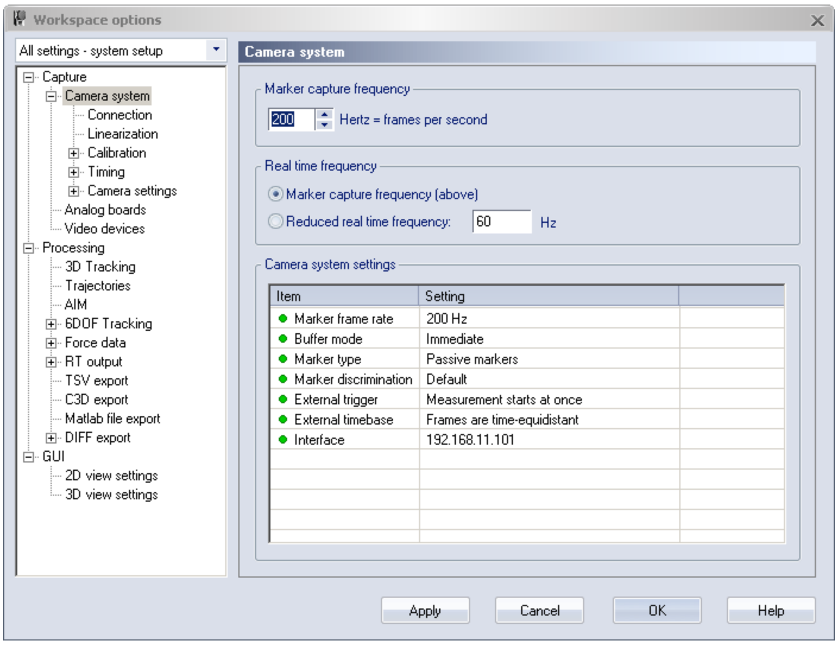
The **Camera system** branch of the options tree consists of hardware settings for the cameras. It has different pages depending on your camera type.

Oqus:
Connection, **Linearization**, **Calibration**, **Timing** and **Camera settings**

ProReflex:
Connection, **Linearization**, **Calibration**, **Timing** and **Camera settings**

MacReflex:
Same as ProReflex except **MacReflex**

The **Camera system** page contains settings for **Marker capture frequency**, **RT capture frequency**and a list with the status of the **Camera system settings**,see sections below.



Marker capture frequency

The **Marker capture frequency** is the capture rate that will be used in a marker measurement. The frequency can be set to an integer between 1 Hz (frames per second) up to the maximum frequency of the current camera type. For Oqus cameras the maximum frequency at full field of view is as follows:

Oqus 1-series - 247 Hz

Oqus 3-series - 500 Hz

Oqus 3+-series - 500 Hz (high speed mode 1764 Hz)

Oqus 5-series - 180 Hz

For information about the maximum frequency on a ProReflex MCU see chapter “MCU technical specifications” on page B - 13. In ProReflex the resolution will change with the capture frequency.

1-120 fps - Full FOV and full resolution

121-240 fps - Full FOV and half vertical resolution

241-500 fps - 70% reduction of FOV both horizontal and vertical and half vertical resolution

501-1000 fps - 50% reduction of FOV both horizontal and vertical and half vertical resolution

Real time frequency

The **Real time frequency** is the frequency that is used in real-time mode, which is started directly when you open a new measurement (also called preview mode). For more information on real-time measurements see chapter "Real time in QTM" on page 271. There are two options for the Real time frequency.

Marker capture frequency
With this option the real-time frequency will be the same as set with **Marker capture frequency**. Which means that the cameras will capture at this frequency and QTM will process as many frames as it can. With this setting you can optimize the real-time frequency because QTM is processing while the camera is capturing the next frame.

Reduced real time frequency
On this option the cameras are set to run on a fixed frequency other than the **Marker capture frequency**. It can be useful if you will make a marker capture at a high frequency, then the real-time can be set to a lower rate so that you can view the data better in preview. This frequency cannot be set higher than the **Marker capture frequency** setting. This is to ensure that you are not using a exposure setting that is too high for the real-time frequency.

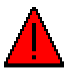
Note: If the analog board uses the **External sync** option, the analog frequency can be too low (less than 20 Hz) when reducing the real time frequency. Because the ratio between the camera frequency and the analog frame rate will still be the same. If this happens increase the real time frequency so that the analog frame rate becomes higher than 20 Hz.

Camera system settings

Under the **Camera system settings** heading, the settings for the current camera system are displayed. Every entry is marked with a notification icon. There are three different notification icons, which have the following meaning:

The specified setting is within the range of the normal settings.

The specified setting is configured in a way that differs from the normal setting. Make sure that it is correct.



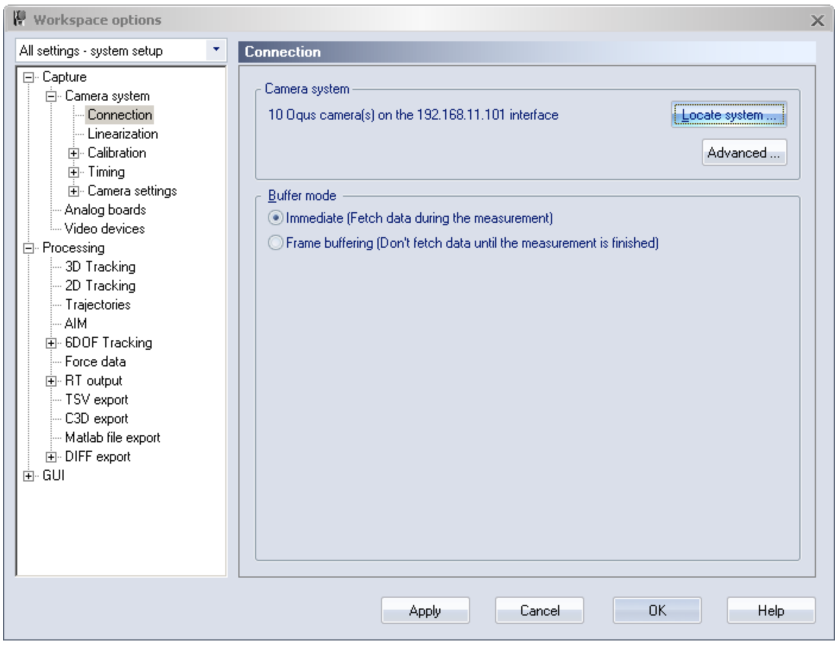
The specified setting is configured in a way that can ruin the measurement if it is not correct. Make absolutely sure that the setting is correct. Some settings may actually damage the camera system, see chapter "External timebase" on page 135.

Note: Oqus cameras uses Frame buffering mode so in that case the yellow triangle is the normal setting.

Click on an entry to go directly to the corresponding page in the **Project options** dialog. By right-clicking on an entry a menu with two options appear: **Change** and

Reset to default(s). Click **Change** to go directly to the corresponding page. Click **Reset to default(s)** to reset that setting to the default value.

Connection



The **Connection** page contains settings for the connection between the camera system and the QTM software. The settings above is how it looks for an Oqus system.

Camera system

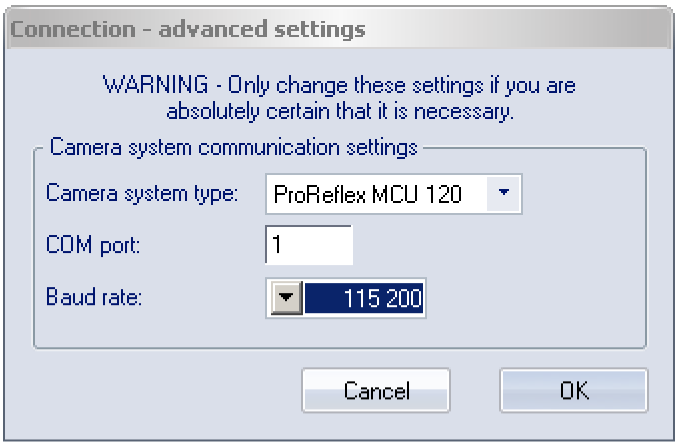
Under the **Camera system** heading there are two possibilities to find the connection to the camera system: automatic connection (**Locate system**) and manual connection (**Advanced**)(**Advanced**).

Advanced

This manual connection option, the **Advanced** option on the **Connection** page, can be used if the automatic connection (**Locate system**) does not work. This option does not work for Oqus cameras.

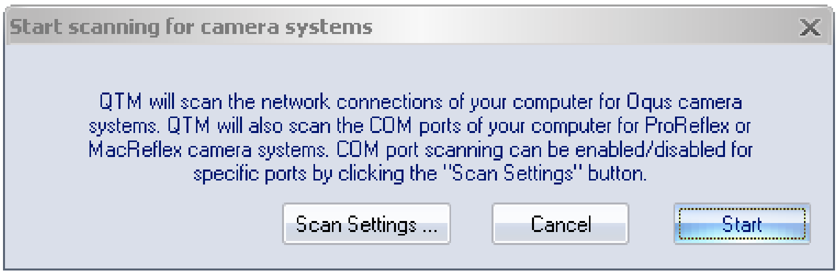
Note: The **Advanced** option should only be used if you are absolutely sure of the correct settings and the automatic scan does not work.

The **Advanced** option will open the **Connection - advanced settings** dialog. In this dialog **Camera system type**, **COM port** and **Baud rate** can be entered manually.



Locate system

Before clicking **Locate system**, make sure that the camera system is switched on. **Locate system** will open a new dialog with three options: **Start**, **Scan Settings** and **Cancel**.



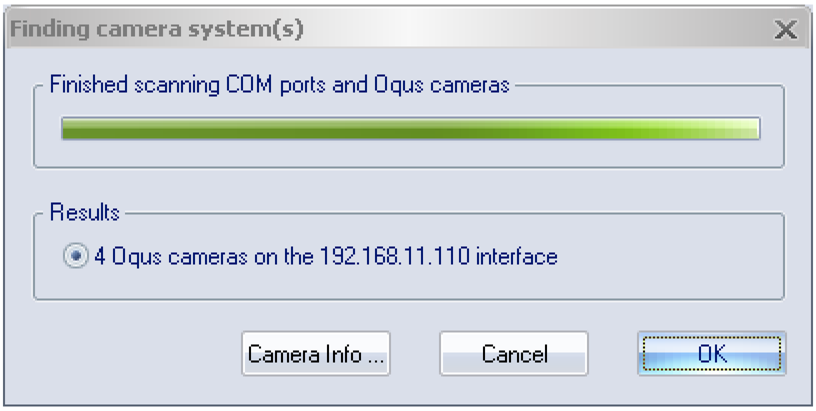
Start

With the **Start** option QTM will automatically search for any camera system that is connected to the Ethernet (Oqus) and COM (ProReflex) ports specified under **Scan Settings**.

Note: If the camera system has old firmware, it will be updated before the system can be used. For more information see chapter "Firmware update when locating system" on page 353.

When QTM has found the camera systems that are connected to the Ethernet and COM ports, it will report the configuration of the camera systems in the **Finding camera system(s)** dialog.

Note: If there are several camera systems, select the camera system that will be used from the list. Only one camera system at a time can be used.



Note: The standard built-in COM port can be used for communication with the camera system. However, it is recommended that either a PCI serial communication board or a PC-Card serial communication board, which both emulate one or more COM ports, is used. These boards run at much higher baud rate than a standard COM port and will therefore speed up the collection of data. Qualisys AB supplies the PCI serial communication board and the PC-Card serial communication board.

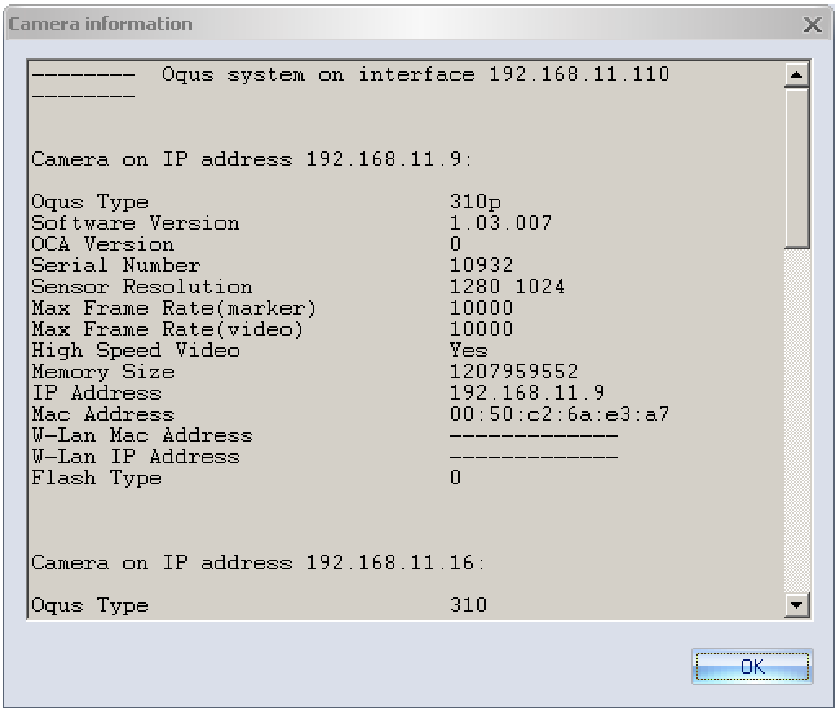
Note: The type of cable is not the same for a standard COM port and a PCI or PC-card serial communication board.

Camera information

When the scanning is finished, it is possible to view some camera information. The **Camera information** dialog is opened with the **Camera info** option on the **Finding camera system(s)** dialog.

For Oqus it shows the information below, which is the same information as **View camera information** on the **2D view window** menu.

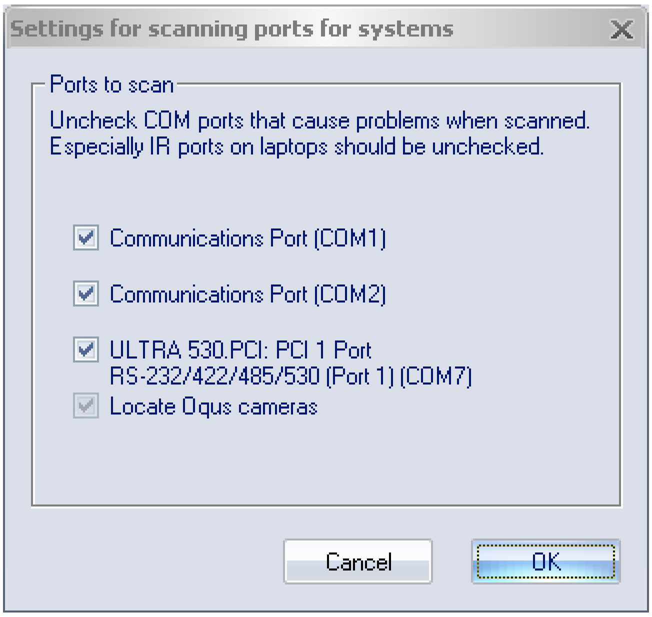
For ProReflex it shows the camera firmware version and other specific data of the camera.



Scan settings

The **Scan settings** option should be used if the scanning process for ProReflex cameras is very slow or malfunctions. It is used to choose which COM ports that are used in the scanning process. QTM will always search for Oqus systems on the available ethernet interfaces.

In the **Settings for scanning ports for systems** dialog, COM ports that are not used for camera systems can be deselected, by clearing the check box next to the port. Especially IR ports may cause problems.



Buffer mode

Under the **Buffer mode** heading on the **Connection** page, the buffering can be set to two modes:**Immediate** and **Frame buffering**.

ProReflex has a third mode called Segment buffering.

Immediate

Immediate mode is the default mode. The motion capture data is then collected by the cameras and stored in each camera's buffer. Simultaneously, QTM requests the stored data as soon as it is available in the cameras. Depending on the number of markers, the capture frequency and the length of the measurement, QTM may spend some time retrieving all the data after the measurement is finished.

Note: For ProReflex this mode cannot be used above 500 Hz. If **Immediate** mode is selected **Frame buffering** mode will be used automatically.

Frame buffering

In **Frame buffering** mode, motion capture data is collected by the cameras and stored in each camera's buffer, just like in **Immediate** mode. QTM, however, will not request the data until the whole measurement is finished.

Oqus
Frame buffering mode can be used when the computer needs more processing time to capture video form DV cameras and it must be used if **Pretrigger** is activated.

ProReflex
For ProReflex this mode means that the measurement can be faster and include more markers than the **Immediate** mode. It can prevent data loss in measurements at high frequencies (500 - 1000 Hz) or when many (or large) markers are used. This is because the cameras have more time for calculations in the **Frame buffering** mode than they have in the **Immediate** mode, since they do not have to send data to the QTM software during the measurement.
Note: Make sure that the whole measurement will fit in the cameras' buffers. For the frame buffer size of the ProReflex MCU see chapter "MCU specifications" on page B - 13. For other MCUs see their manual.

Segment buffering (ProReflex)

Segment buffering is only available for ProReflex cameras and is used for high-speed measurements (over 500 Hz). In this mode the motion capture data is stored in each camera's buffer as segments detected on the CCD. It will save time since the calculations of the markers' X and Y locations are skipped until the whole measurement is finished. After the capture is completed, the calculations of the markers' X and Y location are performed and the measurement computer then retrieves the resulting data. Therefore there no preview of the data can be displayed while capturing. This mode is the fastest and can include more markers than the other modes.

Note: This mode can prevent data loss. It can be used if a high-speed measurement results in lost 2D data, starting from the bottom of the FOV. This mode will, however, result in a shorter measurement time than with the **Frame buffering** mode, but instead a higher number of markers can be captured per frame at high speeds.

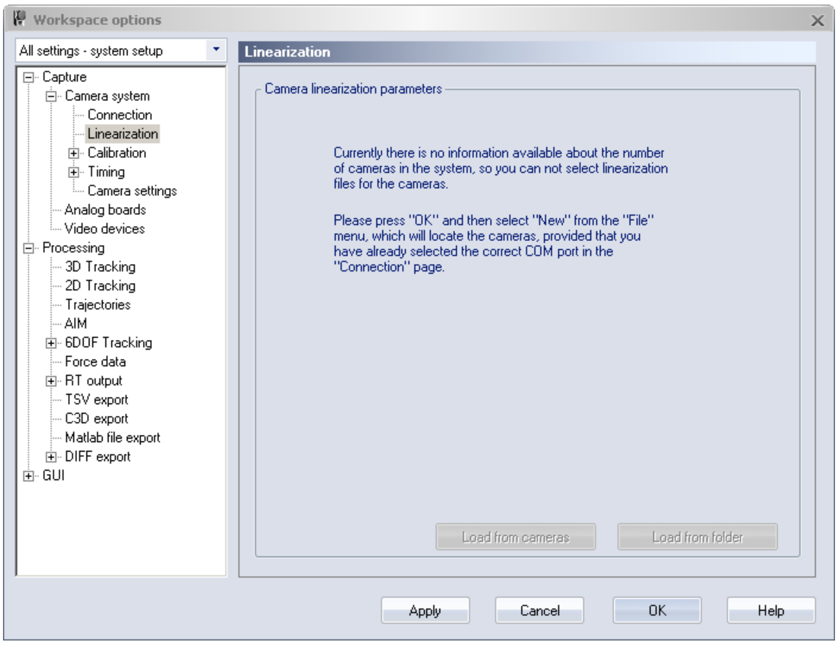
Note: Make sure that the whole measurement will fit in the cameras' buffers. For the segment buffer size of the ProReflex MCU see chapter "MCU specifications" on page B - 13. For other MCUs see their manual.

Linearization

The **Linearization** page contains information about the linearization of each camera, see chapter “Linearization of the cameras” on page 229. Each camera is delivered with its own linearization file, with a file name that includes the serial number of the camera. On this page you can also deactivate cameras in the camera system.

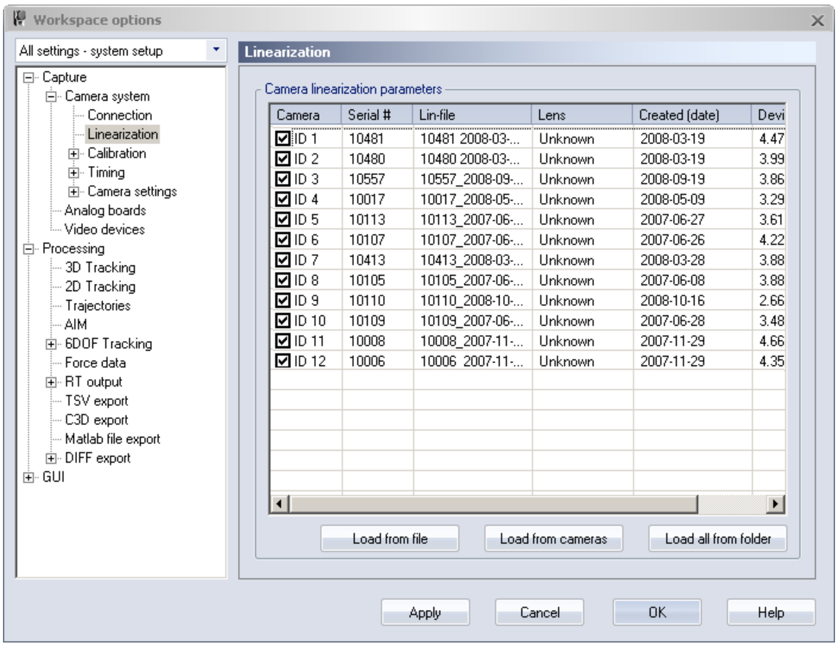
Camera linearization parameters

Under the **Camera linearization parameters** heading there is a list of all the linearization files. However, to be able to select the linearization file for each camera, the QTM software must have been connected to the camera system, see chapter “Connection of the camera system” on page 227. If it has not been connected, the message below is shown.



After the camera system has been connected to the QTM software a list of all cameras appears in the dialog. In the list you can select the linearization files, see instructions below, and select whether a camera will be active in the measurement or not. The camera is active if the checkbox in the **Camera** column is selected. Deselect the checkbox to deactivate the camera.

Note: The camera will still capture data during a measurement even if it is deactivated. Therefore it can be included again in the measurement by reprocessing the file, see chapter “Reprocessing a file” on page 281. However, if a camera has been deactivated during calibration, the calibration must be reprocessed first, see chapter “Reprocess calibration” on page 244.



There are three possible ways to select the camera’s linearization file.

Load from file
Browse to a new file for the selected camera.

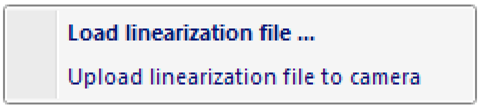
Load from cameras (only Oqus)

Load from folder

Manually from the computer

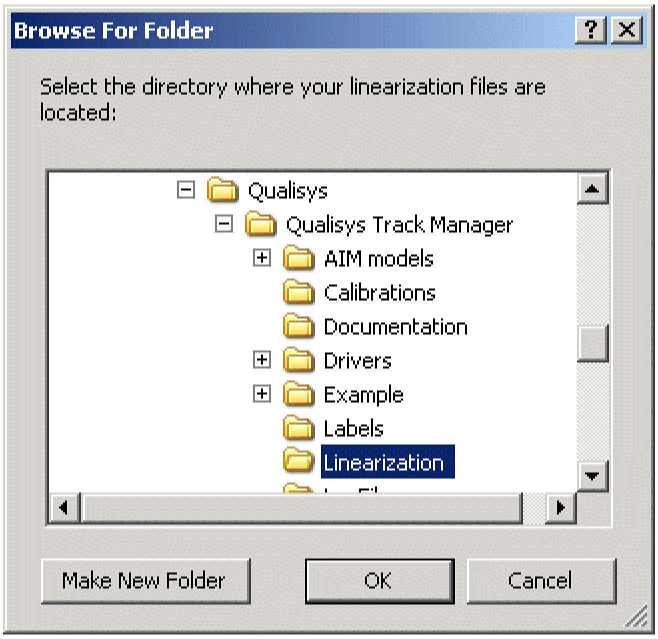
The methods are described below. For Oqus the recommended method is to **Load from cameras**. For ProReflex the recommended method is **Load from folder**.

1. Click on **Load from cameras**. A dialog appears and QTM starts to load the linearization files from the Oqus cameras.
   Note: The file in the camera is the last linearization performed on the camera. However you can also store a specific lin-file in the camera by loading the lin-file manually from the computer and then right-clicking on a camera in the list.

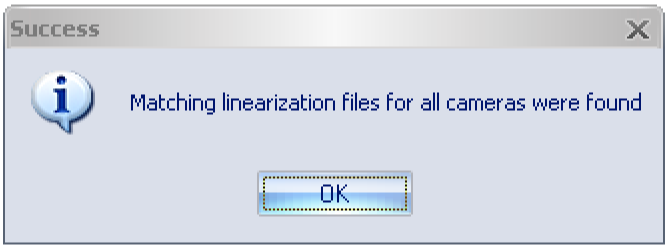
   

   Select **Upload linearization file to camera** to store the current file in the list in the camera. This can be used if you for example performed more than one linearization and don't want to use the last one.

2. Click on **Load from folder**. A dialog appears where you can browse for the folder with the linearization files (*.lin).



If the files have been saved in the **Linearization** folder in the **Qualisys** folder, just click **OK**. Otherwise locate the correct folder and then click **OK**. When you click **OK**, QTM will look for linearization files that match the serial numbers of the cameras and then choose the one with the last date in the file name. If it finds the files in the folder the following dialog is shown and the correct files are installed.



3. Click on the row that shows the camera ID. Browse to the folder where the linearization files (*.lin) are saved and choose the correct file (i.e. the file with the same serial number as the camera).

Calibration

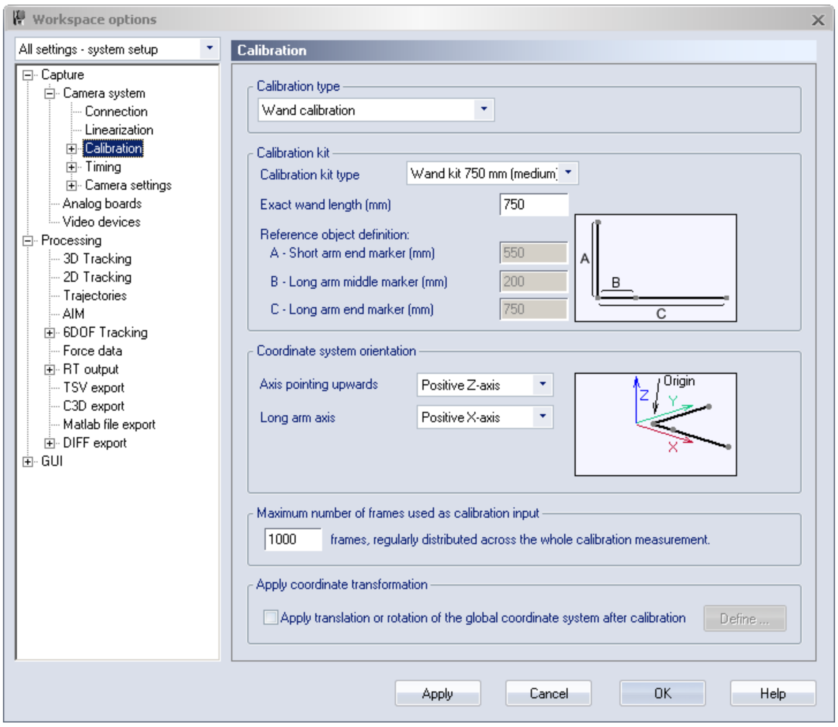
The **Calibration** page contains calibration settings required to perform a correct calibration. It is very important that this data is entered correctly otherwise the motion capture data will not have as high quality as expected. For a detailed description on how to perform a calibration see chapter “Calibration of the camera system” on page 235.

Calibration type

Select the calibration type that will be used. The supported calibration types are**Wand calibration**, **Frame calibration** and **Fixed camera calibration**. All three types are described below.

Wand calibration

Wand calibration uses two calibration objects to calibrate the system. One is a stationary L-shaped reference structure with four markers attached to it. The stationary L-structure (called reference object below) defines the origin and orientation of the coordinate system that is to be used with the camera system. The other calibration object is called calibration wand. It consists of two markers located a fixed distance from each other. This object is moved in the measurement volume to generate data to determine the locations and orientations of the cameras. For more information see chapter “Wand calibration method” on page 241.



Calibration kit

Under the **Calibration kit** heading the calibration kit, which are needed for the calibration, is defined. The calibration kit is used for scaling and locating the coordinate system in the measurement volume. Two objects are needed to calibrate the system: a reference structure and a wand.

Note: The calibration objects are part of the measurement equipment and should be treated with care. A scaling error derived from a damaged calibration object will propagate throughout the whole measurement and analysis.

Calibration kit type

By choosing the **Calibration kit type** in the drop-down box the size of the L-shaped reference structure is specified. The calibration algorithms will then find the reference markers when the calibration recording is made.

The following four settings are available for **Calibration kit type**:

Wand kit 110 mm

Wand kit 300 mm

Wand kit 750 mm

Kit defined below

Exact wand length

Enter the distance between the centers of the reflective markers on the reference wand in **Exact wand length**. It has been measured with high accuracy and can be found on a plate on the wand.

Reference object definition

The **Reference object definition** is only used if **Kit defined below** is selected as the **Calibration kit type**. The positions of the centers of the reflective markers on the L-shaped reference object must then be defined to specify the reference frame markers. The positions are defined as the distance from the corner marker (origin marker) to the other markers, see figure in the **Project options** dialog.

Note: When standard kit types are used the **Reference object definition** will show the distances for the selected kit type.

Coordinate system orientation and translation

Under the **Coordinate system orientation** heading the coordinate system of the motion capture can be customized by choosing the way the X-, Y- and Z-axes are orientated in the measurement volume. The coordinate system of the subsequent motion captures will be the same as that used for the reference structure.

Axis pointing upwards and **Long arm axis** are the settings which decide the directions of the axes. Select the axis that you want for each setting to get the desired coordinate system, see the figure next to the settings to understand how the axes are orientated.

Maximum number of frames used as calibration input

The **Maximum number of frames used as calibration input** setting limits the number of frames used in the calibration process. The default value is 1000 frames. If the number of frames in the calibration file is larger than this setting, the frames will distributed evenly across the whole measurement. E.g. if the calibration time is set to 20 seconds there will usually be 2000 frames in the calibration file. Then with the default number of frames, every second frame will be used to calibrate the system.

Lower this value if you have a volume that you can easily calibrate in about 10-15 seconds. This will make the calculation a bit faster. On the other hand it is good to increase this value if you have a large volume. Especially if the volume is an extended volume where not all of the cameras can see the calibration reference structure. To test this make a long calibration and test with different **Maximum number of frames** to see how it affects your calibration result.

For more information about the calibration time and the frequency that are use see chapter “Calibration dialog” on page 236.

Apply coordinate transformation

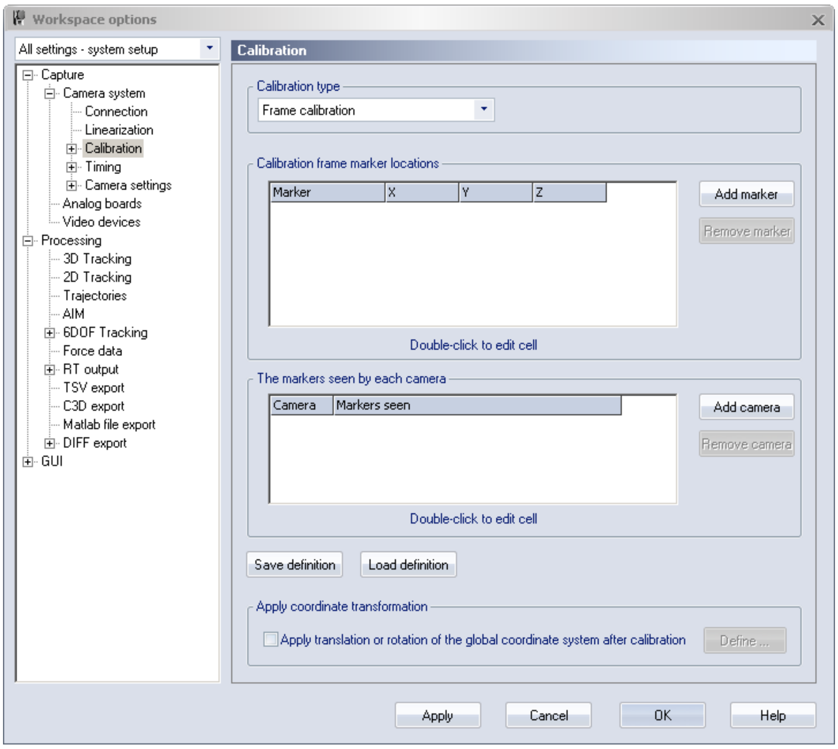
With **Apply coordinate transformation** you can translate and rotate the global coordinate system to any desired position. Select the checkbox and then click **Define** to set the coordinate transformations on the **Transformation** page, see chapter “Transformation” on page 128.

Frame calibration

Frame calibration uses a rigid structure (usually called a calibration frame) with at least five markers attached to it, where the exact location of the markers must be known with as high accuracy as possible. It is recommended that the markers are placed in an asymmetric manner. The origin and orientation of the coordinate system of the motion capture is then determined by the placing of the calibration frame. For more information see chapter "Frame calibration method" on page 243.

Use **Save definition** to save the current frame definition. Load a saved definition with **Load definition**.

Note: The measurements should be performed within the volume that was contained inside the calibration frame. Measurements outside this volume cannot be guaranteed to have a high accuracy.



Calibration frame marker locations

Use the **Add marker** and **Remove marker** options to add or delete markers of the calibration frame. Double-click the X, Y and Z locations of each marker to edit it.

Note: All of the markers must be specified to make a successful Frame calibration.

The markers seen by each camera

Use the **Add camera** or **Remove camera** options to adjust the number of cameras in the system. Enter the numbers of the markers on the calibration frame that are seen by each camera (the numbers are displayed in the **Calibration frame marker locations** list's first column). Separate the numbers with commas.

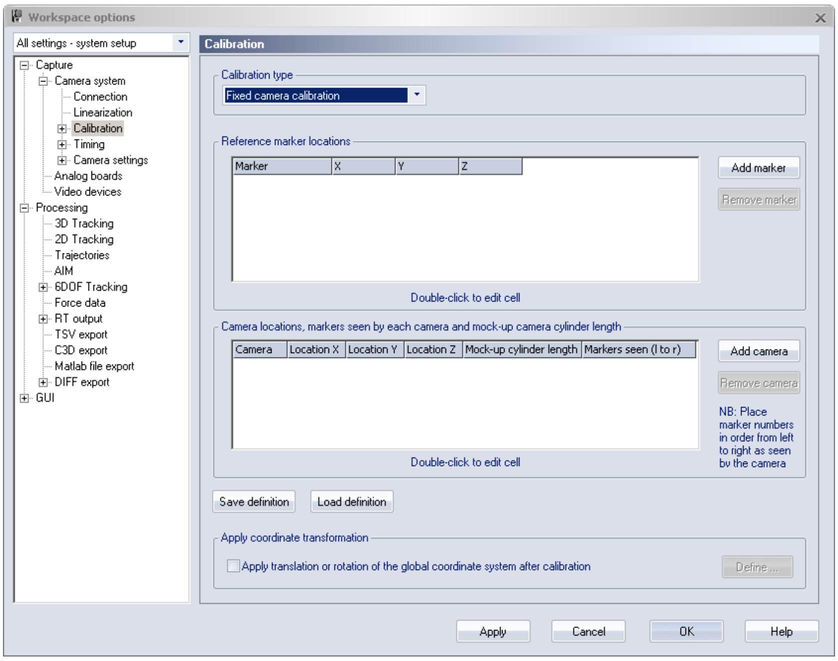
Note: All of the cameras must be entered to make a successful Frame calibration.

Apply coordinate transformation

With **Apply coordinate transformation** you can translate and rotate the global coordinate system to any desired position. Select the checkbox and then click **Define** to set the coordinate transformations on the **Transformation** page, see chapter "Transformation" on page 128.

Fixed camera calibration

On the **Calibration** page for Fixed camera calibration you should enter the data from the survey measurement. If you cannot see this **Calibration** page change the **Calibration type** option to **Fixed camera calibration**. For more detailed information on fixed camera systems contact Qualisys AB about the **QTM - Marine manual**. They include detailed descriptions of the camera installation, survey measurement, fixed camera calibration, validation and the use of 6DOF bodies in marine applications.



Use the options **Save definition** and **Load definition** to save respectively load the data for the Fixed camera calibration. The default folder is the project folder.

Note: The first time you enter the survey data it must be entered manually.

Reference marker locations

Under the **Reference marker locations** heading you should enter the survey data of the reference marker positions. Use the **Add marker** and **Remove marker** options to add or delete reference marker locations. Add the markers in the physical order from left to right. This will make it much easier to enter the markers seen by camera. Double-click the **X**, **Y** and **Z** locations of each marker to edit it.

Note: All of the markers' locations must be entered to make a successful Fixed camera calibration.

Camera locations and markers seen by each camera in order from left to right

Under the **Camera locations and markers seen by each camera in order from left to right** heading you should enter the survey data of the camera positions. Use the **Add camera** to add a new camera last in the list. The cameras must be entered in the same order as the camera system, i.e. the master MCU first and then on through the chain. It is not possible to rearrange the cameras after they have been added, just to remove a camera with **Remove camera**. Double-click the column to enter the following data:

Location X, **Location Y** and **Location Z**
The survey measurement data of the camera.

Mock-up cylinder length
The length of the cylinder that was used on the camera dummy when making the survey measurement.

Note: This length is the horizontal distance between the plate of the camera dummy and the front side of the cylinder. For the ProReflex MCU it is always 18 mm.

Markers seen (l to r)
The markers seen by the camera. Enter them in order from left to right as seen by the camera and separate them with commas (the numbers refer to the first column in the **Reference marker locations** list).

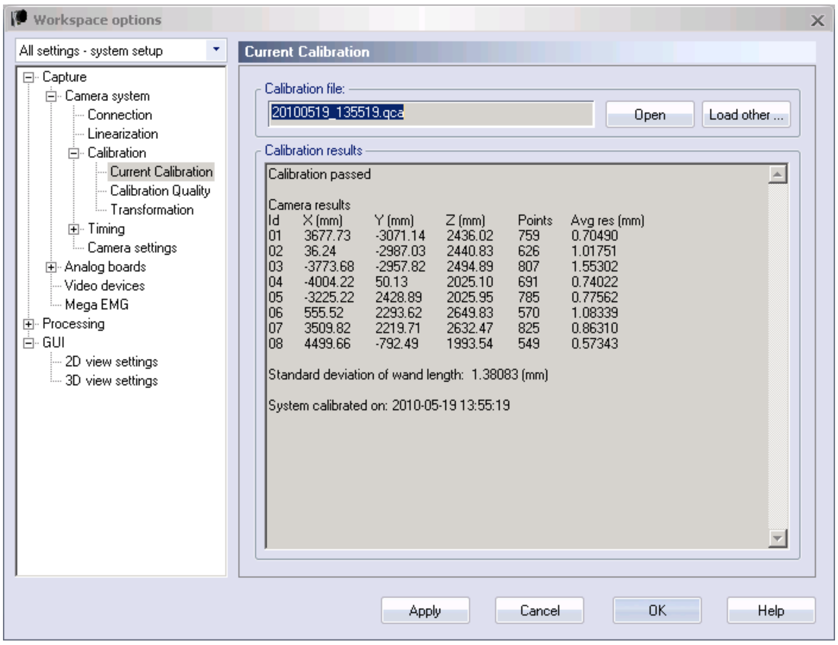
Note: QTM uses the top markers in the **2D view** window as reference markers.

Note: All of the cameras must be entered to make a successful Fixed camera calibration.

Apply coordinate transformation

With **Apply coordinate transformation** you can translate and rotate the global coordinate system to any desired position. Select the checkbox and then click **Define** to set the coordinate transformations on the **Transformation** page, see chapter "Transformation" on page 128.

Current calibration

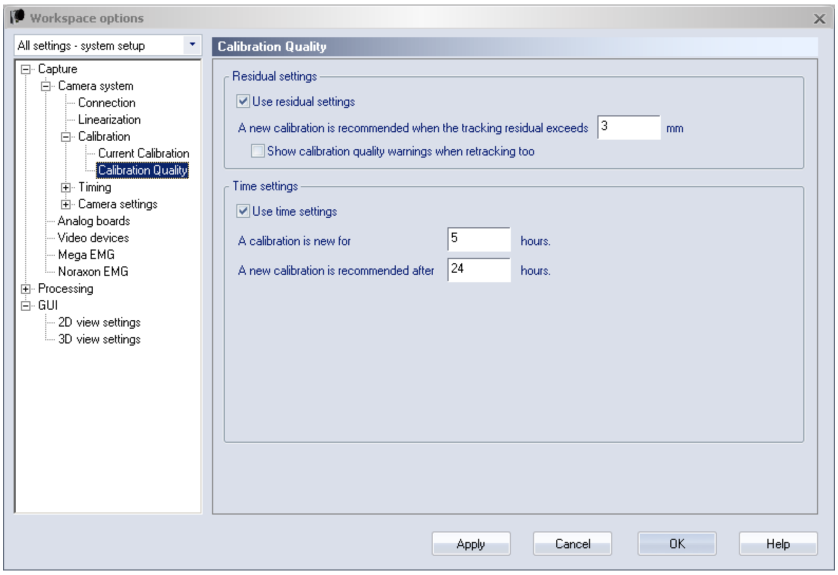


The **Current calibration** page displays the calibration that is used by QTM. Open the current calibration with the **Open** button. With the **Load other** option you can open a dialog and load another calibration file.

If you want to reprocess files with a new calibration, it is often best to change it in the **Reprocessing** or **Batch processing** settings, see chapter "Reprocessing a file" on page 281 and "Batch processing" on page 279.

The **Calibration results** are the same as shown after the calibration, see chapter "Calibration results" on page 237.

Calibration quality



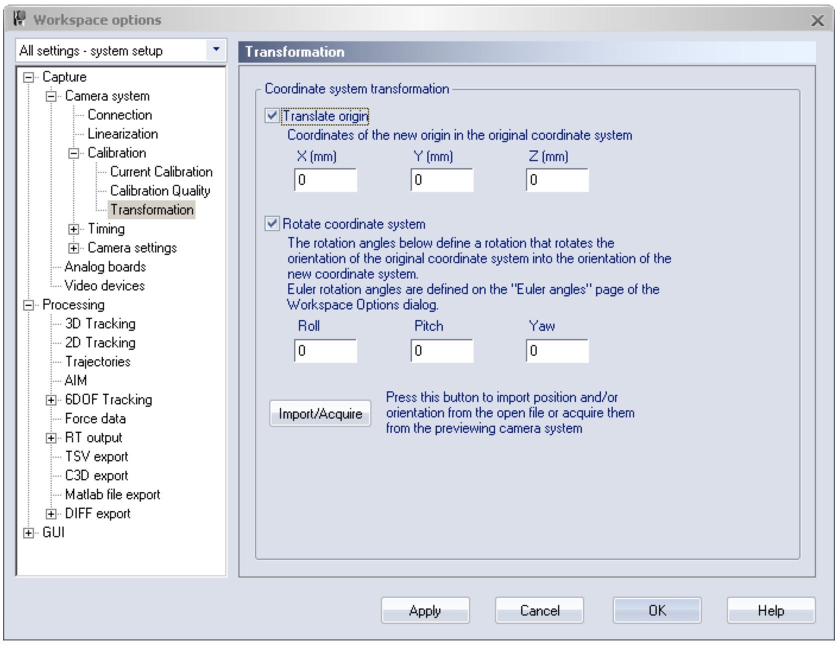
The **Calibration quality** page contains setting for how to detect if the cameras needs to be calibrated. The check can be performed in two different ways.

Residual settings
This will test in the 3D tracker if the cameras have a **Residual** in the **File information** on the **View** menu that is higher than the residual check. The default value for the residual test is 3 mm. It also tests if there are too few of the captured 2D markers used by the 3D tracker, i.e. the number of **Points**. If a camera is considered to be uncalibrated, then there is a warning after the tracking of the file.

Show calibration quality warnings when retracking too
Use this setting if you want to check the calibration quality when you retrack a file.

Time settings
This test will only check how long a calibration is considered as new. The calibration will still be used in a measurement it is just a visual warning. When the time set with **A calibration is new for** has passed the triangle at the bottom right corner turns yellow . Then when the time **A new calibration is recommended after** has passed the triangle changes to orange .

Transformation



The **Transformation** page contains the settings for defining a new global coordinate system. The two changes that can be made to the coordinate system is **Translate origin** and **Rotate coordinate system**. By changing these parameters you can in fact move and turn the coordinate system to any position and orientation. The change is always related to the original position and orientation of the calibration coordinate system. In the case of the Fixed camera calibration this means the origin of the survey measurement.

Important: A new calibration file is saved if you apply a new transformation on the current calibration. The saved calibration only includes calibration data and no 2D data. Therefore you must always go back to the first calibration if you need to recalibrate the calibration.

To activate the change you must select the checkbox of respective setting.

Translate origin (X(mm), Y(mm), Z(mm))
Enter the new position of the origin of the coordinate system (in mm). The direction of the parameters (X, Y and Z) is always related to the original coordinate system of the calibration.

Rotate coordinate system (Roll, Pitch, Yaw)
Enter the new rotation of the coordinate system (in degrees). The rotations always refer to the original coordinate system and they are always applied from left to right. To which axes the rotations are applied depends on the Euler angles definitions on the **Euler angles** page, see chapter “Euler angles” on page 186.

Use **Import/Acquire** to define the rotation from a measured line, see chapter ”Rotate axis to line”.

Note: If the Euler angles are changed after a transformation, the transformed coordinate system will have the same orientation as before but the rotation parameters on the **Transformation** page will change to reflect the new definition.

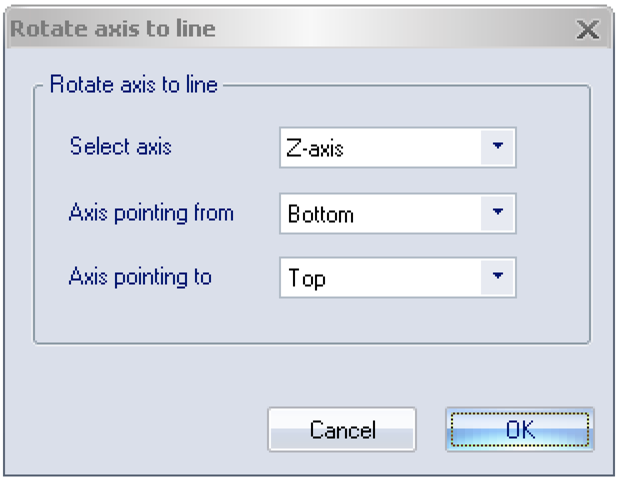
Note: The rotation angles are used to calculate a rotation matrix, which is then used to transform the coordinate system. When the resulting rotation matrix is converted back to rotation angles again it is not necessarily the same angles. This means that the angles that you have entered can change after you have applied it to the coordinate system. E.g. if you just enter a pitch of 100 degrees (using Qualisys standard) this will result in the angles roll = 180, pitch =80, yaw =180.°°°

⚠ **Important:** If you use a custom bounding box on the **3D Tracking** or **6DOF tracking** page or have a force plate, these will be moved with the transformation and the positions must therefore be corrected manually.

Rotate axis to line

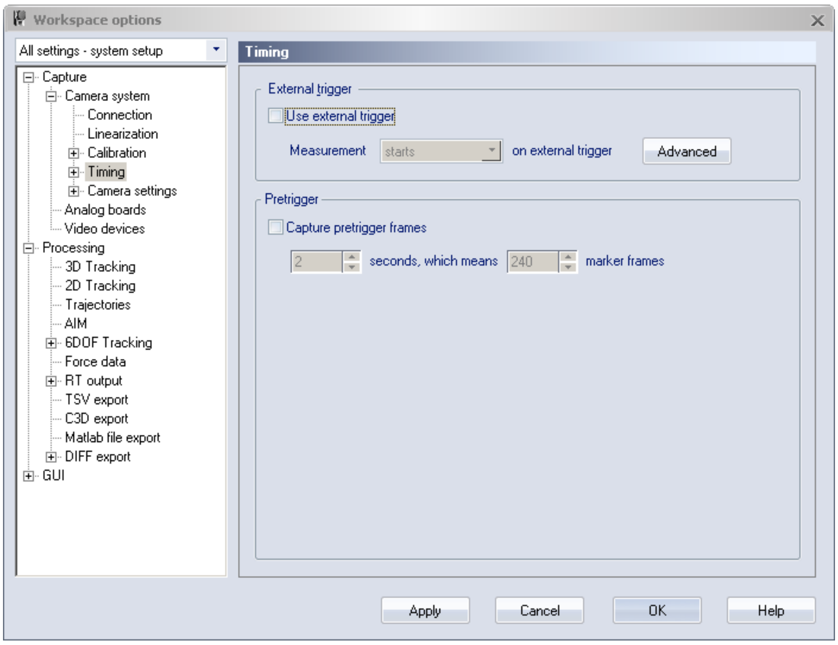
With the **Rotate axis to line** function a line can be used to define the direction of one of the axes. This can for example be useful to define a vertical or horizontal axis if the floor is not level enough. Follow this procedure to use the function:

1. Make a measurement with two static markers that defines the line that you want. It is important that the markers are as static as possible, because an average is used to define the line. It is also important that the file uses the current calibration.
2. Keep the file open and go to the **Transform** page in the **Project options** dialog. If the **Transform** page is not active go to the **Calibration** page and check the **Apply coordinate transformation** box.
3. Activate the **Rotate coordinate system** option on the **Transformation** page.
4. Click on **Import/Acquire** to open the **Rotate axis to line** dialog.



5. Select the axis that you want to define the rotation for.
6. Then select the trajectory that the **Axis is pointing from** and the trajectory that the **Axis pointing to**.
   Note: The position of the coordinate system will not change only the rotation.
   Note: The rotation of the other axes can change after the rotation.
7. Click **OK** to calculate the rotation and show the result on the **Transformation** page.
8. A new calibration file will be saved in the **Calibration** folder after you click OK in the **Project options** dialog.

Timing



The **Timing** page contains the basic timing related settings, these are **External trigger** and **Pretrigger**.

The default setting is that none of the settings are selected. This means that the capture starts as soon as you click **OK** in the **Start capture** dialog or with a delay depending on the **Capture delay**.

For advanced timing settings click the **Advanced** button, see chapter “Advanced (Timing)” on page 132.

External trigger (basic)

With **Use external trigger**, the camera system is started and stopped with an external trigger device. There are therefore three possible settings for **External trigger**:

Starts
The start of a capture is delayed until a hardware trigger event occurs on the **Control** port.

Stops
The capture is stopped by a hardware trigger event at the **Control** port. However the measurement will also stop if the end of measurement time is reached.

Starts and stops
Both the start and stop of the capture is controlled by the trigger event at the **Control** port. For this option there is a **Min time between start and stop** setting on the **Advanced** page.

For advanced trigger settings click the **Advanced** button, see chapter “External trigger (advanced)” on page 133.

Note: For Oqus the trigger can be connected to any camera in the system. For ProReflex the trigger must be attached to the system’s master camera, i.e. the camera connected to the measurement computer. For details on how to connect an external trigger device see chapter “How to use external trigger” on page 375.

Pretrigger (basic)

When selecting **Capture pretrigger frames**, the cameras enter pretrigger mode. In this mode the cameras will continuously write data to the buffer memories of the cameras until a trigger event occurs. The number of frames in buffer is specified in the **Pretrigger** setting and can be specified as a time period in seconds or directly as the number of frames. After the trigger event, the pretrigger frames will be sent to the measurement computer. The trigger event can be sent to the camera either with an external trigger or from a dialog in QTM if there is no external trigger.

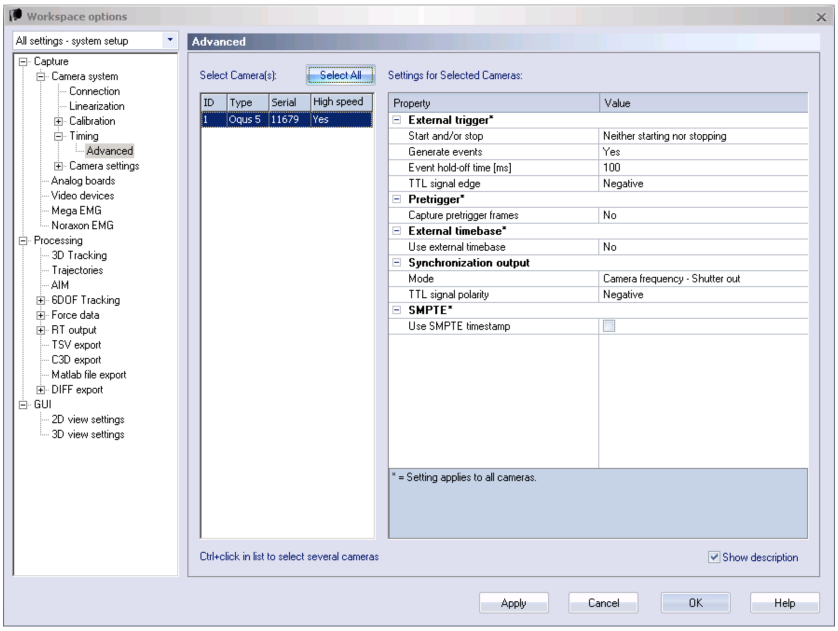
For ProReflex it is important that the total amount of pretrigger data must fit in the camera buffer memory. The size of the data depends on the frame rate and the number of markers. If the buffer memory overflows during the capture an error message will be presented on the ProReflex's front display.

Note: For ProReflex additional hardware is needed if the external trigger is used to start the capture, see chapter "Pretrigger hardware when using external trigger (ProReflex)" on page 377.

Note: Pretrigger cannot be used with segment buffering on the ProReflex.

Warning: Make sure that the time between the start of the capture and the trigger event is long enough to capture all the pretrigger frames, i.e. check that the **Status** bar says "**Waiting for trigger**". Otherwise there will not be enough pretrigger frames to collect from the buffer.

Advanced (Timing)



The **Advanced** page contains all of the timing related settings. There are two lists on the page: to the left there is a list with the cameras in the current camera system and to the right is a list with all the settings for the selected cameras.

The camera list has four columns.

ID
The number of the camera shown in the **2D view** window and on the camera display.

Note: The camera with an M after it is the master camera.

Type
The type of camera: 5-serie, 3+-serie, 3-serie or 1-serie.

Serial
The serial number of the camera.

High speed
Displays whether the camera is a high speed camera or not, so that it is easier to locate the high speed cameras for individual settings.

On the **Advanced** page you can set individual settings for the cameras. Select the cameras that you want to change the settings for in the camera list. You can use Ctrl and Shift to select multiple cameras. The settings list will be updated depending on the cameras that are selected. If multiple cameras are selected and there is a setting that has been set individually its value will say **Differs**. When changing such value it will set all the selected cameras to the same setting.

Use the **Select all** button to select all of the cameras. Only the global settings will be shown in the settings list if none of the cameras are selected.

The settings list contain the settings for the selected cameras. All of the settings marked with * are global. The settings are described in the chapters below. Check the **Show description** option to get a short description of the selected setting.

External trigger (advanced)

External trigger*	
Start and/or stop	Start capture
Generate events	Yes
Event hold-off time [ms]	100
TTL signal edge	Negative
Delay from signal to start/stop [µs]	20000

On the **Advanced** page there are four advanced settings for the external trigger, but you can also control the basic usage from the **Timing** page as well. All of the external trigger settings are global.

Basic settings:

Start and/or stop
Select the external trigger option from the drop-down list.

Neither starting nor stopping
Start the measurement directly when clicking the **Start** button in the **Start capture** dialog. However you can still create events with the external trigger on the Oqus camera.

Start capture
The start of a capture is delayed until a hardware trigger event occurs at the **Control** port of the system's master camera, i.e. the camera connected to the measurement computer.

Stop capture
The capture can be stopped by a hardware trigger event at the **Control** port of the system's master camera. However the measurement will also stop if the end of measurement time is reached.
Note: Events cannot be created on the Oqus camera if you select this option.

Start and stop
Both the start and stop of the capture is controlled by the trigger event at the **Control** port of the system's master camera. For this option a **Minimum time between start and stop** is specified, since the two signals otherwise could be confused, see below.
Note: Events cannot be created on the Oqus camera if you select this option.

Advanced settings:

Min time between start and stop [s]
The minimum time must be between 0.5 s and 1000 s.

Generate events
Activate this option to generate events with the external trigger on Oqus cameras. This option is activated by default.

Event hold-off time
Hold-off time from the last event until a new event can be generated with the external trigger. This is so that there are no extra events caused by contact bounces when releasing the button. The time is by default 100 ms and cannot be longer than 1000 ms.

TTL signal edge
Select the trigger edge to **Positive**, **Negative** or **Any** edge, e.g. pressing a trigger

button is the same as **Negative** edge. **Any** means that any of the two edges will trigger the camera. The setting is for both start/stop and events and is only available with Oqus.

Delay from signal to start [µs]
For Oqus the **Delay signal to start** can be changed. However it must be larger than 20 ms.

Note: ProReflex always starts on negative edge and has a delay of 550 microseconds to the start of capture.

Note: For details on how to connect an external trigger device see chapter "How to use external trigger" on page 375.

Pretrigger (advanced)

Pretrigger*	
Capture pretrigger frames	Yes
Pretrigger time [s]	2.000
Pretrigger frames	200

The settings for pretrigger are the same as on the Timing page, there are no extra settings on the **Advanced** page. All of the external trigger settings are global.

When selecting **Capture pretrigger frames**, the cameras enter pretrigger mode. In this mode the cameras will continuously write data to the buffer memories of the cameras until a trigger event occurs. The number of frames in buffer is specified in the **Pretrigger** setting and can be specified as a time period in seconds (**Pretrigger time**) or directly as the number of frames (**Pretrigger frames**). After the trigger event, the pretrigger frames will be sent to the measurement computer. The trigger event can be sent to the camera either with an external trigger or from a dialog in QTM if there is no external trigger.

For ProReflex it is important that the total amount of pretrigger data must fit in the camera buffer memory. The size of the data depends on the frame rate and the number of markers. If the buffer memory overflows during the capture an error message will be presented on the ProReflex's front display.

Note: For ProReflex additional hardware is needed if the external trigger is used to start the capture, see chapter "Pretrigger hardware when using external trigger (ProReflex)" on page 377.

Note: Pretrigger cannot be used with segment buffering on the ProReflex.

Warning: Make sure that the time between the start of the capture and the trigger event is long enough to capture all the pretrigger frames, i.e. check that the **Status** bar says "**Waiting for trigger**". Otherwise there will not be enough pretrigger frames to collect from the buffer.

External timebase

External timebase*	
Use external timebase	Yes
Signal source	Control port
Signal mode	Periodic
Frequency multiplier	1
Frequency divisor	1
Frequency tolerance in ppm of period time	1000
Control port TTL signal edge	Negative
Delay from signal to shutter opening [µs]	0

Activate the **External timebase** with **Yes** on the **Use external timebase** setting. The external timebase setting applies to all cameras in the system, because it controls the capture rate of the cameras with an external signal.

Note: It is important to use a stable signal source to achieve the best possible synchronization.

For Oqus there are two possible ways to synchronize the external timebase with the camera frames. Either each frame is triggered by a pulse or the camera is locked on to a periodic signal.

Note: On the ProReflex the only option is to trigger each camera frame with an external pulse and therefore the only available options below are **Use external timebase** and **Max expected time between two frames [s]**. The external timebase for ProReflex is only active during the capture, in preview the master camera controls the frame rate.

The following settings can be changed for the external timebase on an Oqus:

Use external timebase
Activate the external timebase with Yes.

Signal source
Oqus can be synchronized with the following signals, where the corresponding settings differ some for the different signals:

Control port
Connect a TTL signal to the Sync in pin of the control port. This is the standard method of synchronizing.

Note: There is no Sync in connection on the standard connector cables for Oqus or ProReflex. For Oqus please contact Qualisys AB for a Control connector with Sync in connection. For ProReflex you can build your own connector from the specifications in chapter “Control port” on page B - 12.

IR receiver
Send an IR signal which is received by the IR receiver on the front of the camera.

SMPTE
Use a SMPTE signal to synchronize. You need to connect an extra synchronization box to convert the signal, see chapter “Using synchronization box for SMPTE” on page 384.

Note: The **Use SMPTE timestamp** option is activated automatically

Signal mode
Select the way to synchronize the camera frame to the signal.

Note: Not available for the **SMPTE** signal.

Periodic
In these modes the camera locks on to the signal of the external timebase and

then the frequency can of the Oqus camera can be set as a multiplier/divisor to get the required frequency for the measurements. This is the recommended setting for periodic signals see chapter “Using External timebase for synchronization to a periodic signal” on page 382.

Non-periodic
In this mode every single frame that is captured must be triggered by a signal from an external timebase source. This means that the frame rate is controlled by the external timebase source. To prevent that the QTM software indicates a timeout error, because no frames are captured, there is a **Max expected time between two frames [s]** setting. Set it to the largest possible interval between two consecutive signals from the external timebase source, for more information see chapter “How to use external timebase” on page 377.

Note: On the Oqus cameras the non-periodic sync cannot be faster than 120 Hz. This is because of network delays described in the **Delay from signal to shutter opening** setting below.

Frequency divisor/multiplier
Set the divisor respectively multiplier for the **Divisor/Multiplier** modes.

Frequency tolerance in ppm of period time
Set the tolerance for the external signal frequency in ppm of the period time. I.e. for the Oqus to lock on to the frequency the period time of the external signal must not differ more than the specified tolerance.

Note: Not available for the **SMPTE** signal.

Control port TTL signal edge
For the **Control port** signal it is possible to specify which edge that will be used as trigger event for the synchronization.

Note: Only available for the **Control port** signal.

Delay from signal to shutter opening
For the **Periodic** modes the delay is 0 by default and can be set to any value.

For the **Non-periodic** the default and minimum time is 19 ms for the Oqus. This is because there must be enough time for the trigger packet to reach all of the cameras in the system. Which means that there is always an delay between the signal and the actual capture. If it is set too short all of the cameras will not receive the TCP/IP message and the synchronization will be lost. To not have to consider the delay set it to the same as the period of your signal. In this way you will not get an image for the very first trigger event but for all the following the capture will be made at the same time as the synchronization signal.

Max expected time between two frames [s]
This setting is only active for the **Non-periodic** mode. Then it decides the maximum time QTM waits for the next frame, i.e. the longest possible time between two signals from the external timebase.

Warning: The camera system can be damaged if the hardware setup is not done properly. Make sure that the resulting camera frequency is not higher than the camera can use in the current mode. For Oqus cameras this means they are limited by the maximum frequency at full image size.

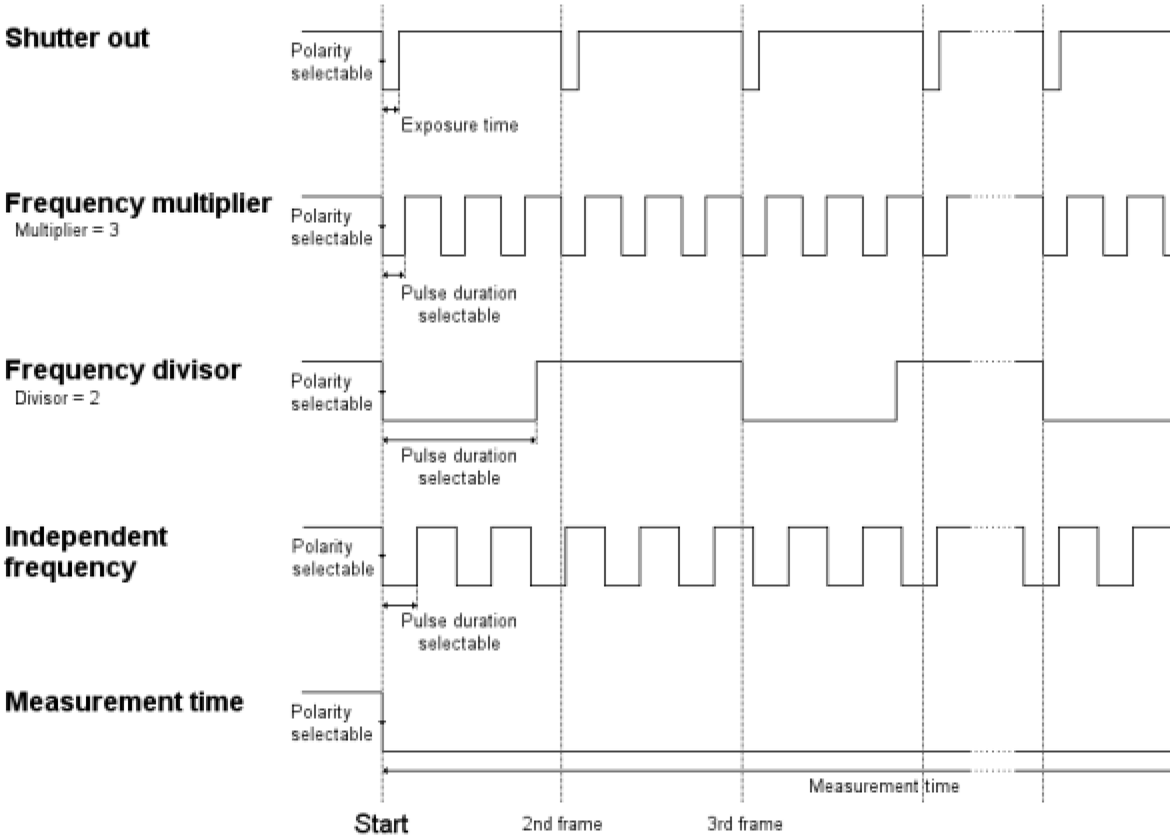
Oqus 1-series - 247 Hz
Oqus 3-series - 500 Hz
Oqus 5-series - 180 Hz

Synchronization output

For Oqus the **Synchronization output** of the camera can be controlled to get a customized output from the camera. This setting can be set independently for each camera so that there can be different outputs in the same system. Select the cameras in the camera list to the left.

Note: If you want to use a sync output signal that is faster than 5000 Hz, then it is recommended that you use the connector on the Master camera. The master camera is displayed with an M next to the ID in the camera list to the left.

There are four different sync modes which are described below. For a description of how to use the sync output see chapter “How to synchronize external hardware” on page 379. The image below is a description of the different sync output settings.



Camera frequency - Shutter out

Synchronization output	
Mode	Camera frequency - Shutter out
TTL signal polarity	Negative

This is the default setting for the camera. In this mode a pulse is sent for each frame captured by the camera and the pulse has the same length as the exposure time.

Set the **TTL signal polarity** to change the polarity of the signal. Negative means that the sync out signal will be low until the first frame when it will go high.

Camera frequency multiplier/Camera frequency divisor

Synchronization output	
Mode	Camera frequency multiplier
Multiplier	3
Duty cycle [%]	50.000
TTL signal polarity	Negative
Camera in marker mode	
Output frequency [Hz]	360.000
Output period time [µs]	2777
Pulse duration [µs]	1389
Camera in video mode	
Output frequency [Hz]	3.000
Output period time [µs]	333333
Pulse duration [µs]	166667

Use this mode to set the frequency as a multiplier/divisor of the camera capture frequency. The multiplication factor/divisor is controlled with the **Multiplier/Divisor** setting. The current **Output frequency [Hz]** and **Output period time [µs]** is shown below the setting both for the camera in marker and video mode, because the marker and video capture rates can be different. The **Output frequency** will change if the camera any of the capture rates are changed. The displayed period time is rounded to the nearest microsecond, but the number of pulses will always be correct compared to the camera frequency.

Note: The maximum **Multiplier** is 1000 and the maximum **Output frequency** is 100000 Hz.

The **Duty cycle** setting for controls how long the pulse will be in percent of the current **Period time**. The actual **Pulse duration** is displayed for the marker and video mode respectively.

Note: The pulse starts at the start of each period, i.e. you cannot apply an offset to the pulse. However by changing the **Duty cycle** and the **TTL signal polarity** you can get an edge at any time between two frames.

Set the **TTL signal polarity** to change the polarity of the signal. Negative means that the sync out signal is high until the start of the first frame when it will go low. Each signal will then be synchronized with its corresponding capture frame which depends on the **Multiplier/Divisor** setting, see image above.

Camera independent frequency

Synchronization output	
Mode	Camera independent frequency
Output frequency [1-100000Hz]	100
Duty cycle [%]	50.000
TTL signal polarity	Negative
Output period time [µs]	10000
Pulse duration [µs]	5000

Use this mode to set the frequency independently of the camera capture frequency. The **Output frequency** can be set between 1 and 100000 Hz. When changing the frequency the **Output period time [µs]** will then update to its corresponding value. However the **Output period time** is displayed in microseconds, so it is important to notice that a small change in the period time will not always change the displayed **Output frequency**. But the camera will always send the correct number of output pulses in relation to its capture frequency.

The **Duty cycle** setting for controls how long the pulse will be in percent of the current **Period time**. The actual **Pulse duration** is displayed below.

Note: The pulse starts at the start of each period, i.e. you cannot apply an offset to the pulse. However by changing the **Duty cycle** and the **TTL signal polarity** you can get an edge at any time between two frames.

Set the **TTL signal polarity** to change the polarity of the signal. Negative means that the sync out signal is high until the start of the first frame when it will go low. The first signal will always be synchronized with the camera capture, the following will signals will be dependent of the relation between the camera capture rate and the **Output frequency**.

Measurement time

⊟ Synchronization output	
Mode	Measurement time
TTL signal polarity	Negative

Use this mode to get a pulse that last for the whole measurement time. The output will go low/high at the start of the measurement and not go high/low until the end of the last frame.

Set the **TTL signal polarity** to change the polarity of the signal. Negative means that the sync out signal is high until the start of the first frame when it will go low.

SMPTE

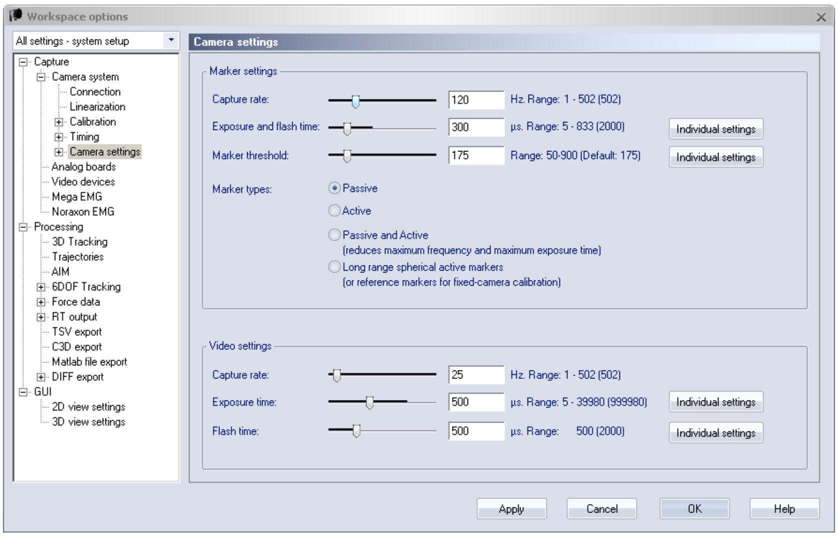
⊟ SMPTE	
Use SMPTE timestamp	☑

For Oqus an SMPTE timestamp signal can be connected to the camera and then displayed in QTM. However the connection requires the synchronization box to convert the signal so that it can be measured by the camera, see chapter “Using synchronization box for SMPTE” on page 384.

Select the **Use SMPTE timestamp** option to activate the collection of the timestamp on the system. When a measurement is started the cameras will automatically detect which one the SMPTE signal is connected to. The timestamp is displayed in the **Timeline control** bar see chapter “Timeline control bar” on page 61.

Note: A trigger button cannot be connected to the camera with the SMPTE signal, because it uses the Trig in pin of the control port.

Camera settings (Oqus)



The **Camera settings** page contains the basic settings for the Oqus camera. All of the settings on this page are global, i.e. the settings is applied on all cameras in the camera system. For individual camera settings go to the **Advanced** page or click on one of the **Individual settings** button.

Note: When mixing the different types of Oqus cameras the limits of global settings will be set to the lowest value of the different types.

Marker settings

The **Marker settings** apply to all Oqus cameras that are in marker mode, i.e. calculating markers in the **2D view** window. The settings can be changed either with a scroll bar or by entering the number directly. The current possible range is displayed next to the setting. The number within parenthesis is the maximum value that can be set for the camera types in the system, if the range is lower than the maximum you need to change another setting to increase the range. For example set the **Capture rate** lower to increase the **Exposure and flash time** range.

The following settings are available on the **Camera settings** page. For tips on how to set **Exposure time** and **Marker threshold** see chapter "Tips on marker settings in QTM" on page 250. For more marker settings go to the **Advanced** page.

Note: When mixing the different types of Oqus cameras, the global settings are limited to the lowest value for the different types.

Capture rate

The **Capture rate** is the frequency that is used during a marker measurement. It is the same setting as **Marker capture frequency** on the **Camera system** page, which means that if the setting is changed it is updated on both pages.

The possible capture range is shown left of the setting. The range is changed depending on the Oqus type and also the size that is set with **Image size** and the current capture range is shown next to the capture rate. In the **Project options** dialog the **Image size** must be changed first to get a higher frequency. However in the **Camera settings sidebar** the **Image size** will be reduced automatically with a higher frequency. The following ranges are used for full size images:

Oqus 1-serie - 1-247 Hz

Oqus 3-serie - 1-500 Hz

Oqus 3+-series - 500 Hz (high speed mode 1764 Hz)

4-series - 485 Hz

Oqus 5-serie - 1-187 Hz

Exposure and flash time
The **Exposure time** setting changes the exposure for marker mode. Increase this setting if the markers are not visible. Decrease the exposure if you have extra reflections. The **Range** shown to the left of the setting is the range that can be used with the current capture rate. The maximum value is a tenth of the period (1/capture rate). The exposure time is also limited to 2 ms in marker mode, because that is the maximum flash time. The current limit of the setting is shown with a blue bar in the scrollbar.

For Oqus this setting is the main option to change amount of light in the image. Because then you can change the light input without touching the cameras. Especially for Oqus 3- and 5-series it is good to have the aperture maximum opened and change the exposure time instead.

Note: This setting can be set individually, click on **Individual settings** to go to the **Advanced** page. The setting will say **Differs** if the setting has been set individually.

Marker edge threshold level
The **Marker edge threshold level** sets the threshold for markers in Oqus cameras, the default value is 175. Every pixel value that is brighter than this value will be calculated as an marker.

When you lower the threshold more pixels will be considered to be a marker. If you set the threshold too low there might be strange effects, including no markers or very long markers. On the other hand a too high threshold can also mean that the camera cannot calculate any markers.

Note: This setting can be set individually, click on **Individual settings** to go to the **Advanced** page. The setting will say **Differs** if the setting has been set individually.

Marker types
Select which marker type to use.

Passive
The passive marker is the default marker type and it works with any marker with reflective material.

Active
The active marker is the smaller active marker with sequential coding.

Passive and Active
In this mode the camera will capture both passive and active markers. This mode can be used if you need to add some temporary markers to the subject and do not want to add active markers. If you mix the passive and active markers all the time you loose some of the advantages of both types.

Long range spherical active markers
Any of the long range active marker and reference marker also the old 40 mm active marker.

Video settings

The **Video settings** apply to all Oqus cameras that are in video mode, i.e. showing a video image in the **2D view** window. The settings can be changed either with a scrollbar or by entering the number directly. The current possible range is displayed next to the setting. The number within parenthesis is the maximum value that can be set for the camera types in the system, if the range is lower than the maximum you need to change another setting to increase the range. For example set the **Capture rate** lower to increase the **Exposure time** range.

The following settings are available in the **Camera settings** page.

Capture rate
The **Capture rate** under the **Video settings** heading is only used for video capture and can therefore be set independently of the marker capture rate. The maximum capture rate is decided by the **Image size** and the current capture range is shown next to the capture rate. In the **Project options** dialog the **Image size** must be changed first to get a higher frequency. However in the **Camera settings sidebar** the **Image size** will be reduced automatically with a higher frequency. The following ranges are used for full size images:

Oqus 1-serie - 1-247 Hz

Oqus 3-serie - 1-500 Hz

Oqus 3+-series - 500 Hz (high speed mode 1764 Hz)

Oqus 5-serie - 1-180 Hz

Note: It is possible to capture video with Oqus cameras that are not high-speed. However then the capture rate is limited to 30 Hz, even if you have set a higher frequency on the **Video** page.

Note: The 4-series is not available with a high-speed video configuration.

Exposure time
Sets the **Exposure time** in microseconds. The maximum exposure time is 20 microsecond less than the period time, i.e. 1/capture rate. The current limit of the setting is shown with a blue bar in the scrollbar.

Note: This setting can be set individually, click on **Individual settings** to go to the **Advanced** page. The setting will say **Differs** if the setting has been set individually.

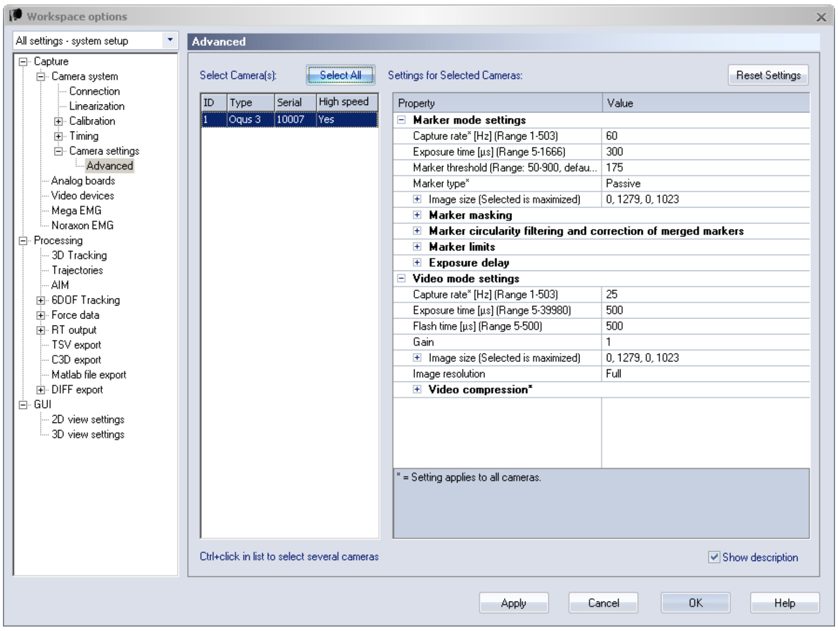
Flash time
Sets the **Flash time** in microsecond. The maximum flash time is always limited by the exposure time but there are also other limitations. First of all the flash time cannot be longer than 2 ms. The other limitation is that it can only be one tenth of the period time of the capture rate. For example if the capture rate is 200 Hz the maximum flash time is 500 us. The current limit of the setting is shown with a blue bar in the scrollbar.
The IR flash does not help much when you capture video. Therefore it is often a good idea to set the **Flash time** to 0.

Note: This setting can be set individually, click on **Individual settings** to go to the **Advanced** page. The setting will say **Differs** if the setting has been set individually.

For more video settings go to the **Advanced** page.

Note: When mixing the different types of Oqus cameras, the global settings are limited to the lowest value for the different types.

Advanced (Oqus)



The **Advanced** page contains all of the camera settings for the Oqus camera. There are two lists on the page: to the left there is a list with the cameras in the current camera system and to the right is a list with all the settings for the selected cameras.

The camera list has four columns.

ID
The number of the camera shown in the **2D view** window and on the camera display.
Note: The camera with an M after it is the master camera.

Type
The type of camera: 5-serie, 3-serie, 3+-serie or 1-serie.

Serial
The serial number of the camera.

High speed
Displays whether the camera is a high speed camera or not, so that it is easier to locate the high speed cameras for individual settings.

On the **Advanced** page you can set individual settings for the cameras. Select the cameras that you want to change the settings for in the camera list. You can use Ctrl and Shift to select multiple cameras. The settings list will be updated depending on the cameras that are selected. If multiple cameras are selected and there is a setting that has been set individually its value will say **Differs**. When changing such value it will set all the selected cameras to the same setting.

Use the **Select all** button to select all of the cameras. Only the global settings will be shown in the settings list if none of the cameras are selected.

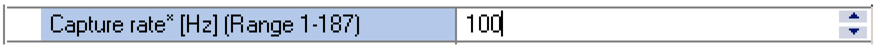
The settings list contain the settings for the selected cameras. All of the settings marked with * are global. The settings are described in the chapters below. Check the **Show description** option to get a short description of the selected setting.

Use the **Reset settings** button to reset all of the camera settings to the default values.

Note: When mixing the different types of Oqus cameras, all global settings will be limited to the lowest value of camera type limits. However when the settings are set individually it is possible to set the value within the camera type limits.

The **Marker mode settings** contains all of the settings for marker mode. The settings are described below.

Capture rate



The **Capture rate** is the frequency that is used during a marker measurement. The setting is global for all cameras.

It is the same setting as **Marker capture frequency** on the **Camera system** page, which means that if the setting is changed it is updated on both pages.

The possible capture range is shown to left of the setting. The range is changed depending on the Oqus type and also the size that is set with **Image size**. In the **Project options** dialog the **Image size** must be changed first to get a higher frequency. However in the **Camera settings sidebar** the **Image size** will be reduced automatically with a higher frequency. The following ranges are used for full size images:

Oqus 1-serie - 1-247 Hz

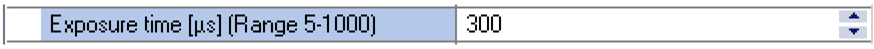
Oqus 3-serie - 1-500 Hz

Oqus 3+-series - 500 Hz (high speed mode 1764 Hz)

4-series - 485 Hz

Oqus 5-serie - 1-187 Hz

Exposure time



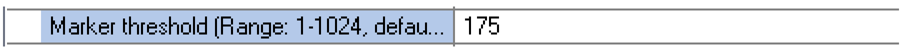
The **Exposure time** setting changes the exposure time for marker mode. For marker mode the exposure time and flash time is the same because it is no meaning exposing the image longer than the flash. The setting can be set individually for each camera.

Increase this setting if the markers are not visible. Decrease the exposure if you have extra reflections. The **Range** shown to the left of the setting is the range that can be used with the current capture rate. The maximum value is a tenth of the period (1/capture rate). The exposure time is also limited to 2 ms in marker mode, because that is the maximum flash time.

For Oqus this setting is the main option to change amount of light in the image. Because then you can change the light input without touching the cameras. Especially for Oqus 3- and 5-series it is good to have the aperture maximum opened and change the exposure time instead.

For tips on how to set the exposure see chapter “Tips on marker settings in QTM” on page 250.

Marker threshold



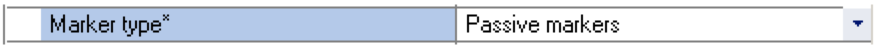
The **Marker threshold** sets the threshold intensity level for markers in Oqus cameras. The setting can be set individually for each camera.

The level can be set between 1 and 1024, the default value is 175. Every pixel value that is brighter than the specified value will be calculated as an marker.

When you lower the threshold more pixels will be considered to be a marker. If you set the threshold too low there might be strange effects, including no markers or very long markers. On the other hand a too high threshold can also mean that the camera cannot calculate any markers.

For tips on how to set the exposure see chapter "Tips on marker settings in QTM" on page 250.

Marker type

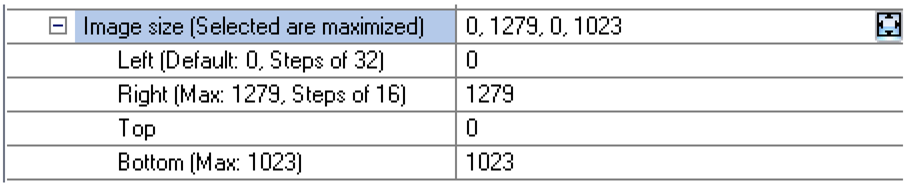


Click on for the **Marker type** setting to select the marker type for the cameras. The setting is global for all cameras. There are three options for this setting:

> **Passive markers**
> Any marker with reflective material
>
> **Long range active markers / Reference markers**
> Any of the long range active marker and reference marker also the old 40 mm active marker.
>
> **Close range active markers**
> The close range active marker also called flat active marker.

A passive marker reflects the IR light from the camera flash, while an active marker transmits IR light. However, if continuous active markers are used the marker type should be set to **Passive markers**.

Image size



With **Image size** it is possible to change the pixels that are active when you capture marker data. The setting can be set individually for each camera. Use the maximize button to reset the selected cameras to the maximum image size.

The reason for changing the size is to be able to capture at a higher frequency. The frequency range will be updated automatically when you change the size. The image size is showed with an red rectangle in the preview **2D view** window. The cameras will still capture markers outside the rectangle in preview, but when you make a measurement that data is discarded.

The image size is specified in **Left** and **Right**, **Top** and **Bottom**. Where the **Left** and **Right** values can only be specified in certain steps depending on the Oqus series. The steps for the current model is displayed next to the settings. The maximum size depends on the Oqus series according to the table below.

Oqus	Max Right	Max Bottom
1-series	639	479
3-series	1279	1023
3+-series	1295	1023

3+-series (high speed mode)	648	512
4-series	1696	1710
5-series	2351	1727

Note: The x size of the 3-serie camera must be at least 160 pixels. The x size of the 3+-serie camera must be at least 144 pixels in **Standard** mode and 72 pixels in **High-speed** mode.

Note: The 16 extra pixels of the 3+-series are not active pixels and will be displayed as a black band, if you zoom in on the right side of the image.

Note: The image size of the 5-series camera can only be changed in the y-direction. This is because the frame rate on this sensor will not be increased when changing the number of pixels in the x-direction.

Marker masking

Marker masking	
Number of masks	1
Auto marker masking	Press the button to add masks
Mask 1	239, 1271, 0, 343
Left (x-min)	239
Right (x-max)	1271
Top (y-min)	0
Bottom (y-max)	343
Use current masks in measurements	Yes

With the **Marker masking** option masks can be added in an Oqus camera where no markers will be calculated. The masks will be shown as green areas in the **Marker** and **Marker intensity** mode. For information on how to use the marker masking see chapter “How to use marker masking” on page 264.

Use the ... button on the **Auto marker masking** option to automatically add markers masks to the selected cameras. You can select several cameras when applying the auto marker mask, but it is important to remove any real markers from the volume. The cameras must be in RT/Preview mode to use this feature. For more information see chapter “How to use auto marker masking” on page 265.

The marker mask is set individually for each camera. Select one camera and add a mask by clicking the plus button on the **Number of masks** line. There can be a maximum of 5 masks per camera. Use the minus button on the line to remove a mask. Modify the values for the mask in the list.

Left (x-min)
The start pixel of the masking area in X-direction in camera coordinates.

Right (x-max)
The end pixel of the masking area in X-direction in camera coordinates.

Top (y-min)
The start pixel of the masking area in Y-direction in camera coordinates.

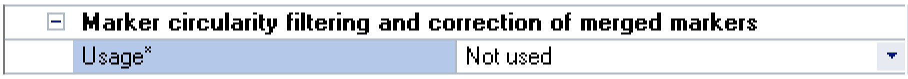
Bottom (y-max)
The end pixel of the masking area in X-direction in camera coordinates.

Note: The origin of the camera coordinates is the upper left corner of the 2D view and X-direction is horizontal and Y-direction is vertical.

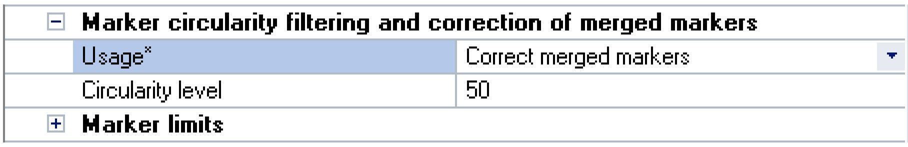
Note: The pixels can be converted to subpixels by a multiplication of 64.

Use current masks in measurements
Enable the use of the marker masks in the measurements. Use this option to turn off the masks to check that the masks are correct. The masks will be grey in the **Marker** and **Marker intensity** mode and you can see the markers behind the mask.

Marker circularity filtering and correction of merged markers



The **Marker circularity filtering and correction of merged markers** options are used for filtering and correction of the 2D data. The default is that the filtering and correction is turned off (**Not used**). For more information about the filtering see chapter “How to use circularity filtering and correction of merged markers” on page 267.



The marker filtering is done in the camera according to the options below.

Usage
To turn on the filtering and correction select the option that you want on the **Usage** option. Which markers that are filtered out depends on the **Circularity level** option. The markers are color coded in the 2D view depending on the filter, see chapter “How to use circularity filtering and correction of merged markers” on page 267.

Not used
Use no filtering of the marker data.

Correct merged markers
The markers filtered out by the **Circularity level** setting will be tested for whether they are merged or not. All merged markers that are detected will be corrected in to two separate markers.
The other markers that were filtered out but not considered to be merged will be kept. However if there are more than two other cameras seeing the marker, the filtered out marker is not used by the 3D tracker.

Correct merged markers, remove others
The same as **Correct merged markers** setting, but all the markers that were not considered to be merged will be removed in the 3D tracker.

Remove all non-circular markers
All markers that are filtered out by the **Circularity level** setting will be removed in the 3D tracker.

Circularity level
The **Circularity level** option defines which markers are filtered out as too non-circular. These markers will then processed according to the **Usage** setting.
The option is in percent, meaning the relation between x and y size. The default value is 50. When set to 0 all markers are considered to be non-circular. A level of 100 means that all markers are considered to be circular.
Note: There can be too many markers considered as non-circular in the camera. In that case there is a red warning in the left upper corner of the 2D view and all the markers that the camera have not been able to check are considered to be OK.

Marker limits

The **Marker limits** settings decides which reflections that are detected as markers by the cameras. When **Use default settings** is set to **True** no marker limits are used and QTM will detect all reflections as markers.

Marker discrimination	
Use default settings	False
Marker size	128, 60000
Smallest	128
Largest	60000
Max number of markers per frame	10000

Set the **Use default settings** to **False** to set other **Marker limits** settings. The setting can be set individually for each camera.

The **Marker limits** settings will affect the 2D data when it is captured. When the file has been captured the 2D data cannot be manipulated in any way. There are three discrimination settings:

- **Marker size**
  - **Smallest**
  - **Largest**
- **Max number of markers per frame**

Note: The **Smallest** and **Largest** parameters are **not** in mm, they are in subpixels, i.e. the sizes depend on the distance between marker and camera. Check the 2D data in the **Data info** window to see the size of the markers you want to remove.

Smallest

The **Smallest** parameter controls the smallest detectable size of a marker for each camera in every frame. The size of the marker is given in subpixels. Any marker which is smaller than is specified in this parameter will be discarded from the camera output.

This option might be useful to screen out tiny non-marker reflections or to assure a minimum size of the markers seen by the camera.

The minimum value of this parameter is 1 and the maximum is the value of the **Largest** parameter.

Largest

The **Largest** parameter controls the largest detectable size of a marker for each camera in every frame. The size of the marker is given in subpixels. Any marker larger than what is specified in this parameter will be discarded from the camera output.

This option might be useful to screen out reflections of other cameras, which tend to be large. It is also useful on other large non-marker reflections.

The minimum value of this parameter is the value of the **Smallest** parameter and the maximum is 60000.

Max number of markers per frame

The **Max number of markers per frame** parameter controls the maximum number of markers transmitted by each camera on a per frame basis. The value can be specified with any integer between 1 and 10000.

When there are more markers in a frame than the value of this parameter, some will be skipped. Since markers are calculated from the top of the image and downward, markers will be skipped from the bottom of the camera's FOV. For example, if there are 10 real markers within the FOV of a camera and the **Max number of markers per frame** parameter has been set to 5 markers, it is just the 5 topmost markers that will be calculated and transmitted to the measurement computer.

Exposure delay

Exposure delay	
Exposure delay mode*	Camera group
Camera groups	Camera group 1

The **Exposure delay** setting shifts the exposure of a camera compared to other cameras in the system. This can be used when the flash of a camera disturbs another camera in the system. However because the time of the exposure will then be different in the camera system there can be tracking problems, for more information see chapter “Delayed exposure to reduce reflections from other cameras” on page 252.

Note: Delayed exposure only works for Oqus 1- and 3-series. This is because the 5-series camera has a rolling shutter, which means that some part of the image is always being exposed.

To activate the exposure delay select **Camera group** or **Advanced** on the **Exposure delay mode** setting. When the delayed exposure is activated **Using delayed exposure** is displayed in the Status bar. The delayed exposure setting is displayed in each camera's 2D view, e.g. expgrp: 1 when the camera is in exposure group 1.

The recommended mode is **Camera group** because then the delay is calculated automatically. First select the cameras that you want in the group from the camera list to the left. Then select one of the five groups in the **Camera groups** setting. E.g. if you only have two groups you should use group 1 and 2. QTM will then automatically delay the exposure time of the groups so that the delay for a group is the same as the longest exposure time in the group before. This means that you can use any exposure time for the cameras and the delay will always be correct.

Exposure delay	
Exposure delay mode*	Advanced
Delay time [µs] (Range 0-19700)	0

The **Advanced** mode should only be used if you are completely sure about what you are doing. The delay is then set with an absolute time in the **Delay time** setting. The delay time will be the same even if you change the exposure time of a camera in the system, which means that you must then manually change the **Delay time** for the cameras to keep the correct delays to reduce reflections.

Sensor mode

Sensor Mode	Standard

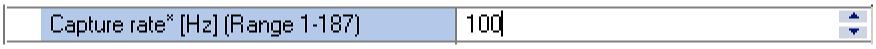
The **Sensor mode** option is only available on the 3+-series. It changes the resolution of the sensor between **Standard** (1296*1024) and **High speed** (648*512). In **High speed** mode you can therefore capture at 1746 Hz with the same FOV as the **Standard** mode. The setting can be changed individually on each camera and also for Marker and Video mode separately.

The sensor mode setting can be changed on all versions of the 3+ cameras, it has nothing to do with the High speed video version of the Oqus cameras.

Video mode settings

The **Video mode settings** contains all of the settings for video mode. The settings are described below.

Capture rate



The **Capture rate** on the **Video** page is only used for video capture and can therefore be set independently of the marker capture rate. The setting is global for all cameras.

The maximum capture rate is decided by the **Image size** and the current capture range is shown next to the capture rate. In the **Project options** dialog the **Image size** must be changed first to get a higher frequency. However in the **Camera settings sidebar** the **Image size** will be reduced automatically with a higher frequency. The following ranges are used for full size images:

Oqus 1-serie - 1-250 Hz

Oqus 3-serie - 1-500 Hz

Oqus 3+-series - 500 Hz (high speed mode 1764 Hz)

Oqus 5-serie - 1-187 Hz

Note: It is possible to capture video with Oqus cameras that are not high-speed. However then the capture rate is limited to 30 Hz, even if you have set a higher frequency on the **Video** page.

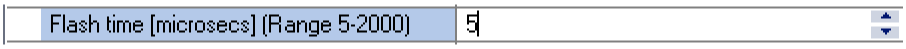
Note: The 4-series is not available with a high-speed video configuration.

Exposure time

The **Exposure time** setting changes the exposure time for video mode. The setting can be set individually for each camera.

The setting sets the exposure time in microseconds. The maximum exposure time is 20 microsecond less than the period time, i.e. 1/capture rate.

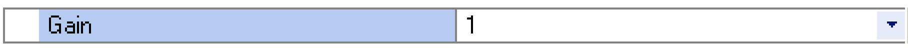
Flash time



The **Flash time** setting changes the flash time for video mode. The setting can be set individually for each camera.

The setting sets the flash time in microsecond. The maximum flash time is always limited by the exposure time but there are also other limitations. First of all the flash time cannot be longer than 2 ms. The other limitation is that it can only be one tenth of the period time of the capture rate. For example if the capture rate is 200 Hz the maximum flash time is 500 us.
The IR flash does not help much when you capture video. Therefore it is often a good idea to set the **Flash time** to 5.

Gain

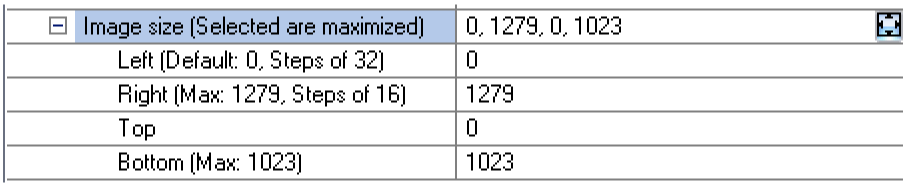


With the **Gain** option the intensity of the video image can be increased. Select the desired gain from the drop-down list. The options are 1, 2 and 4. Use the **Gain** option if the exposure time and aperture is already at their maximums.

The 3- and 5- series has digital gain, which means that the intensity of the image is just multiplied with the desired gain. The result is that image quality and number of grey scales are decreased.

The 3+- and 1-series has analog gain before the image capture and therefore the image quality is not decreased at the same degree with increasing gain. The number of grey scales in the image is also the same for different gain settings.

Image size



With **Image size** it is possible to change which part of the image that is captured in video mode. The setting can be set individually for each camera. Use the maximize button to reset the selected cameras to the maximum image size.

The selected image size will be shown with a red square in the video preview window. The frequency range will be updated automatically when you change the size. The camera will still capture video outside the rectangle in preview, but when you make a measurement the image is cropped.

The image size is specified in **Left** and **Right**, **Top** and **Bottom**. Where the **Left** and **Right** values can only be specified in certain steps depending on the Oqus series. The steps for the current model is displayed next to the settings. The maximum size depends on the Oqus series according to the table below.

Oqus	Max Right	Max Bottom
1-series	639	479
3-series	1279	1023
3+-series	1279	1023
3+-series (high speed mode)	648	512
4-series	1696	1710
5-series	2351	1727

Note: The x size of the 3-serie camera must be at least 160 pixels. The x size of the 3+-serie camera must be at least 144 pixels in **Standard** mode and 72 pixels in **High-speed** mode.

Note: The image size of the 5-serie camera can only be changed in the y-direction. This is because the frame rate on this sensor will not be increased when changing the number of pixels in the x-direction.

Image resolution

The Oqus image resolution will change the quality of the images that are captured by Oqus according to the options below. The setting can be set individually for each camera.

Note: To optimize the memory usage there is a column of black pixels to the right in the image. Which is especially visible at 1/64 or lower resolution.

Resolution
Set the number of pixels that are captured, e.g. with 1/4 resolution every other

line and column is deleted. Choose the resolution for the camera from the options below.

Full - Use all of the pixels

1/4 - Use every second pixel in x and y direction

1/9 - Use every third pixel in x and y direction

1/16 - Use every fourth pixel in x and y direction

1/36 - Use every sixth pixel in x and y direction

1/64 - Use every 8th pixel in x and y direction

1/256 - Use every 16th pixel in x and y direction

Video compression

Video compression*	
Compression codec	<None>
Max cameras to fetch from simultane...	2
Codecs recommended by Qualisys	Press the button to follow the link

The **Video compression** option can be used to reduce the video file size, by default it is set to **<None>** for uncompressed image and maximum quality. The settings applies to all cameras.

The **Max cameras to fetch from simultaneously** setting specifies how many cameras are fetched from at the same time after the capture. The default number of cameras are 0 which means fetch from unlimited number of cameras. Change it to 2 if you have problem with the fetching of video data, which almost maximizes the 100 Mbit bandwidth of a daisy-chained system. You can increase the number if the system is connected in a star configuration with a Gigabit ethernet switch.

Video compression*	
Compression codec	FFDS
Configure codec	Press the button for configuration
Max cameras to fetch from simultane...	2
Codecs recommended by Qualisys	Press the button to follow the link

Choose a codec from the list on the **Compression codec** line to activate it. The codecs in the list are the ones that are installed on your computer.

The option **<None>** will save the video file uncompressed. If the file is uncompressed it will not play at the correct speed in any program. This is because the amount of data is too large for graphic board to handle.

The codecs are then grouped in **Recommended codecs** and **Other codecs**. The recommended codecs are **FFDS** for smaller files with lossy compression and **Lagarith** for larger files with lossless compression. For more information about these codecs see chapter “Codecs for Oqus high-speed video files” on page 261.

Note: If none of these codecs are installed on the computer there are no alternatives under the **Recommended codecs** heading. Download and install the codecs according to the description.

The settings for a codec are opened with the ... button on the **Configure codec** line. The settings depend on the codec, please refer to the codec instructions for help on the settings. The FFDS codec is recommended with the MJPEG codec and it is important to set the settings according to the chapter “FFDS-MJPEG codec settings” on next page.

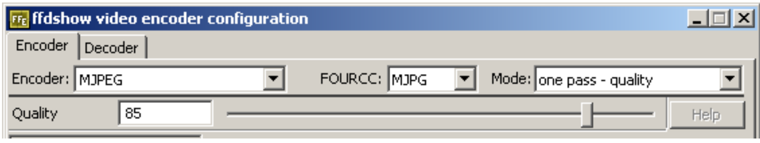
Some codecs have a quality setting that controls the compression, where 100 % is the best quality and also the largest files. With **Use default compression quality** set to **Yes**

QTM uses the default of the codec, which is not the same as the best quality. Set **Use default compression quality** to **No** to activate the **Compression quality** setting to set the quality manually.

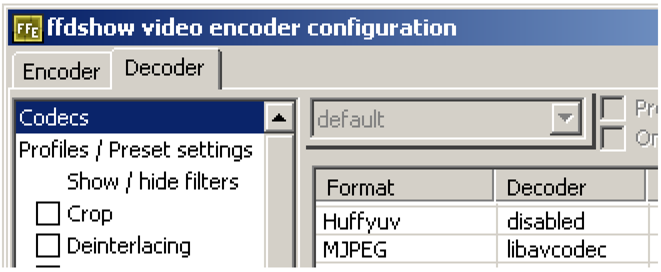
Click on the ... button on the **Codecs recommended by Qualisys** line to go to a website with links to codecs.

FFDS-MJPEG codec settings

The **FFDS** codec is a package of many codecs. We recommend the **MJPEG** codec (called Encoder in the codec settings) and a **Quality** of at least 85 (change the **Mode** to 'one pass - quality' in the codec settings). Higher quality means less compression and therefore higher quality.



It is important to change the **Decoder** setting for **MJPEG** to **libavcodec**, otherwise the files will not be played in QTM. Click on the **Decoder** tab and then **Codecs** in the list to the right. Scroll down to find the MJPEG codec.



Color interpolation algorithm

The **Color interpolation algorithm** option is only available if you have connected an Oqus camera with a color sensor. For the color cameras you can select between the following options for the color interpolation.

Note: It is important to notice that the color interpolation is time consuming and therefore the RT frequency will be much lower with color interpolation.

Note: In a measurement the color interpolation is done after the video has been fetched from the camera, so it does not change the maximum time that you can capture video.

Note: If you want to use the color interpolation with the **Image resolution** option, then you must select 1/9 or 1/36 for the color interpolation to work. The other options deletes the color information from the sensor.

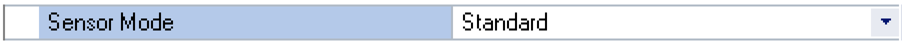
No interpolation
Capture the raw black and white image. The image will be checkered because of the color filter in front of the sensor. This pattern cannot be removed because the filter is integrated with the sensor.

Bilinear interpolation
The fastest color interpolation, but also with slightly lower quality than the other two options.

VNG interpolation
Slower color interpolation, but also higher quality than bilinear interpolation.

AHD interpolation
The most time consuming interpolation, but also with the highest quality.

Sensor mode



The **Sensor mode** option is only available on the 3+-series. It changes the resolution of the sensor between **Standard** (1296*1024) and **High speed** (648*512). In **High speed** mode you can therefore capture at 1746 Hz with the same FOV as the **Standard** mode. The setting can be changed individually on each camera and also for Marker and Video mode separately.

The sensor mode setting can be changed on all versions of the 3+ cameras, it has nothing to do with the High speed video version of the Oqus cameras.

Active filtering

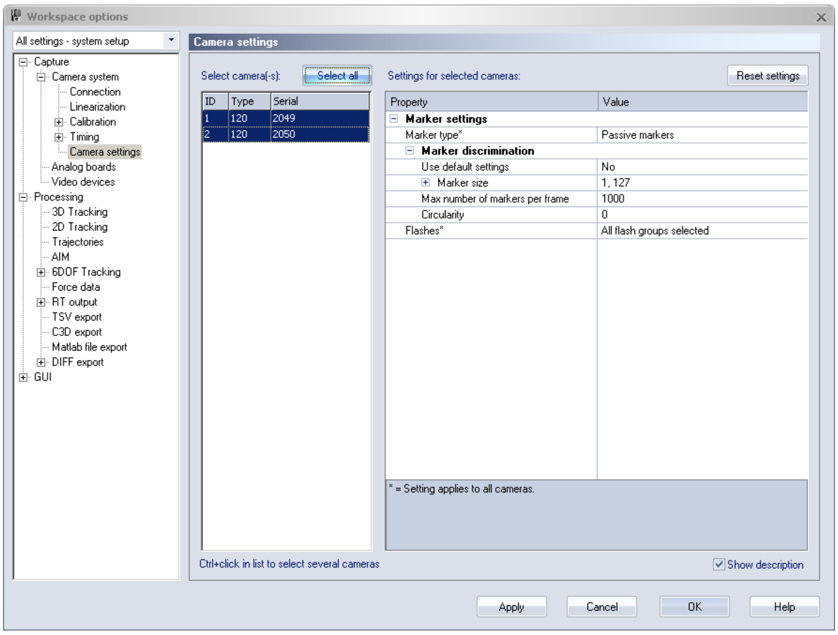
The **Active filtering** mode is used to improve capturing in daylight conditions. It is available on the 4-, 3+- and 1-series cameras and if activated it is used both in marker and video mode. The **Active filtering** mode can be activated individually on each camera. However usually it is best to activate it on all cameras in the system, but for example if one camera is capturing high-speed video then you should deactivate Active filtering on that camera. For more information about active filtering see chapter “How to use active filtering” on page 264.

There are two settings for the **Active filtering** mode:

Continuous (Recommended)
The **Continuous** setting is the default and recommended filtering setting. When activated the camera captures an extra image before the actual image. With the exact same settings as the actual image except that there is no IR flash. The extra image is then subtracted from the actual image so that the background light is suppressed. This helps in for example daylight conditions where the background light is much brighter than indoors.
Note: The maximum capture rate is limited to about 50% of the cameras normal maximum capture rate when **Active filtering** is activated. This is because there are two images captured for every frame. For example on the 3+-series the maximum frame rate is 240 Hz at full image size.

Single
With the **Single** setting the camera only captures an image before starting a measurement. As for the **Continuous** settings the extra image uses no IR flash but is otherwise the same as the actual image. This setting can be used if you have a very static setup.

Camera settings (ProReflex)



The **Camera settings** page contains the camera settings for the ProReflex camera. There are two lists on the page: to the left there is a list with the cameras in the current camera system and to the right is a list with all the settings for the selected cameras.

The camera list has four columns.

ID
The number of the camera shown in the **2D view** window.

Type
The type of camera: 120, 240, 500, 100.

Serial
The serial number of the camera.

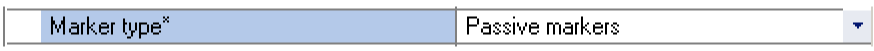
High speed
Displays whether the camera is a high speed camera or not, so that it is easier to locate the high speed cameras for individual settings.

You can set individual **Marker discrimination** settings for the cameras. Select the cameras that you want to change the settings for in the camera list. You can use Ctrl and Shift to select multiple cameras. The settings list will be updated depending on the cameras that are selected. If multiple cameras are selected and there is a setting that has been set individually its value will say **Differs**. When changing such value it will set all the selected cameras to the same setting.

Use the **Select all** button to select all of the cameras. Only the global settings will be shown in the settings list if none of the cameras are selected.

The settings list contain the settings for the selected cameras. All of the settings marked with * are global. Check the **Show description** option to get a short description of the selected setting. The settings are described in the chapters below.

Marker type



Click on for the **Marker type** setting to select the marker type for the cameras. The setting is global for all cameras. There are three options for this setting:

Passive markers
Any marker with reflective material

Long range active markers / Reference markers
Any of the long range active marker and reference marker also the old 40 mm active marker.

Close range active markers
The close range active marker also called flat active marker.

A passive marker reflects the IR light from the camera flash, while an active marker transmits IR light. However, if continuous active markers are used the marker type should be set to **Passive markers**.

Marker discrimination

The **Marker discrimination** settings decides which reflections that are detected as markers by the cameras. When **Use default settings** is set to **True** no marker discriminations are used and QTM will detect all reflections as markers.

⊟ **Marker discrimination**	
Use default settings	False
⊟ Marker size	1, 127
Smallest	1
Largest	127
Max number of markers per frame	1000
Circularity	0

Set the **Use default settings** to **False** to set other **Marker discrimination** settings. The setting can be set individually for each camera.

The **Marker discrimination** settings will affect the 2D data when it is captured. When the file has been captured the 2D data cannot be manipulated in any way. There are three discrimination settings:

Marker size

Smallest

Largest

Max number of markers per frame

Circularity

Note: The **Smallest** and **Largest** parameters are **not** in mm, they are in per mill of the diagonal of the CCD, i.e. the sizes depend on the distance between marker and camera. E.g. if 19 mm marker is used they should **not** be set to 15 respectively 25, to see which sizes to use instead check the 2D data in the **Data info** window.

Largest

The **Largest** parameter controls the largest detectable size of a marker for each camera in every frame. The size of the marker is given in per mill of the diagonal of the CCD. Any marker larger than what is specified in this parameter will be discarded from the camera output.

This option might be useful to screen out reflections of other cameras, which tend to be large. It is also useful on other large non-marker reflections.

The minimum value of this parameter is the value of the **Smallest** parameter and the maximum is 127.

Smallest

The **Smallest** parameter controls the smallest detectable size of a marker for each camera in every frame. The size of the marker is given in per mill of the diagonal of the FOV. Any marker which is smaller than is specified in this parameter will be discarded from the camera output.

This option might be useful to screen out tiny non-marker reflections or to assure a minimum size of the markers seen by the camera.

The minimum value of this parameter is 1 and the maximum is the value of the **Largest** parameter.

Maximum number of markers per frame

The **Maximum number of markers per frame** parameter controls the maximum number of markers transmitted by each camera on a per frame basis. The value can be specified with any integer between 1 and 1000.

When there are more markers in a frame than the value of this parameter, some will be skipped. Since markers are calculated from the top of the image and downward, markers will be skipped from the bottom of the camera's FOV. For example, if there are 10 real markers within the FOV of a camera and the **Maximum number of markers per frame** parameter has been set to 5 markers, it is just the 5 topmost markers that will be calculated and transmitted to the measurement computer.

Note: The number of markers in the camera output can be limited, even if it is within the limit of the parameter. One reason for deleted markers is for example too high frame rate.

Circularity

The **Circularity** parameter controls how round the markers must be to be detected as markers. The parameter can be used to delete markers that are partly hidden or merged with other markers. This option will increase the accuracy of the system, but at the same time lead to more deletions in the camera output if markers are somewhat distorted.

The value is set to the difference between marker extension in the X and Y directions expressed as a percentage between 0 and 100. 0 is the default value, which means that all markers are detected regardless of their circularity. 100 % means that the circularity has to be nearly perfect.

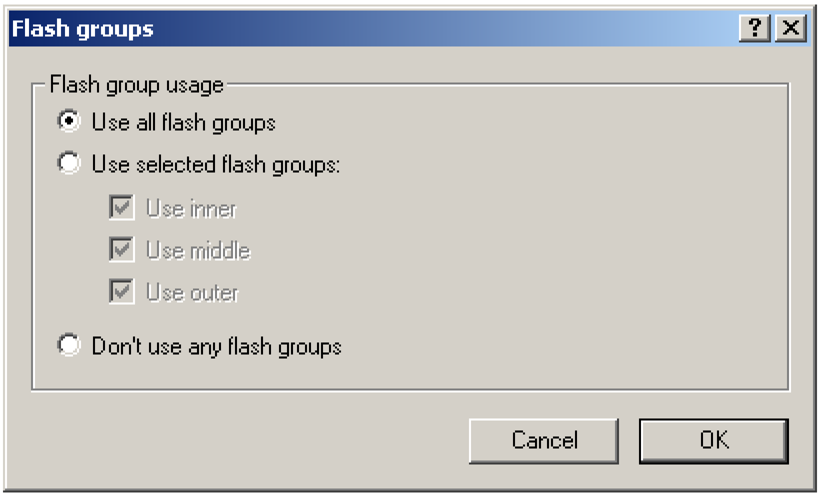
For example, if the **Circularity** is set to 80, the extension in the X direction must be between 80% and 125% of the extension in the Y direction to be accepted as a marker. I.e. if the Y size is 5 points the X size must be between 4 and 6 points (actually 6.25) to be accepted.

Flashes

The **Flashes** setting controls the IR flashes from the cameras. The flashes are used to light up passive reflective markers or to activate active markers. Click the button ... to the right to change the settings in the **Flash groups** dialog.

Note: These settings are only valid for ProReflex MCUs.

Flash groups



In the **Flash groups** dialog there are options for the flash groups of the MCU.

There are three flash groups in the ProReflex MCU: outer, middle and inner group. Activating the different flash groups will adjust the flash intensity and the strobe volume, so that the camera is optimized for different distances. The following selections of flash groups are recommended with respect to the distance between cameras and markers:

Distance	Flash group
0 – 2.5 m	Inner
0 – 4 m	Inner + Middle
0 – 30 m	Inner + Middle + Outer

The **Use all flash groups** option activates all three flash groups in all cameras.

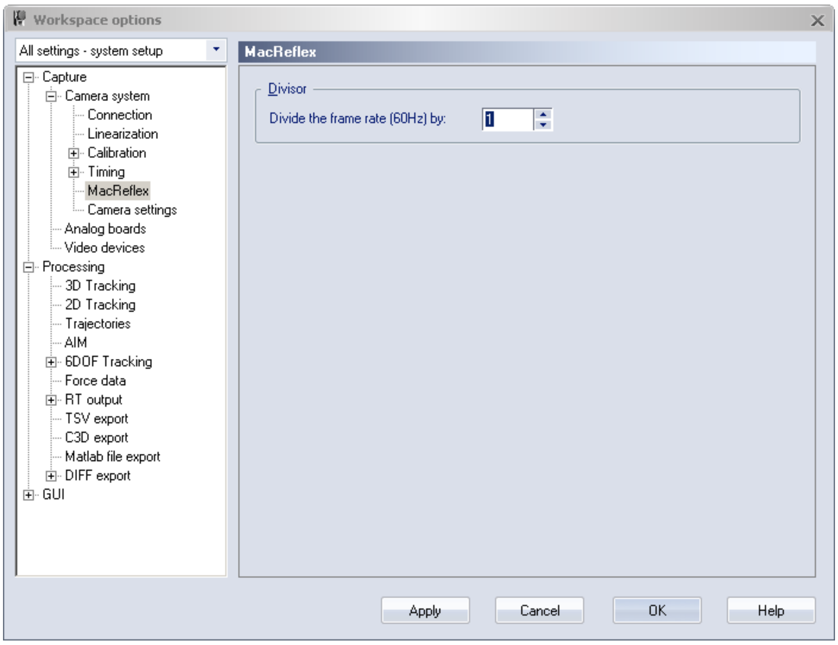
With the **Use selected flash groups** option, the different flash groups can be activated from a list.

The **Don’t use any flash groups** option deactivates all the flash groups. It can be used with active markers that work without control from the camera’s flashes.

MacReflex

The **MacReflex** page is only available if a MacReflex camera system is connected to QTM. The page contains a setting for the division of the frame rate. The MacReflex camera uses a frame rate of 60 Hz, which with the **Divisor** setting can be divided by an integer. E.g. if the setting is set to 3, the frame rate will be 20 Hz.

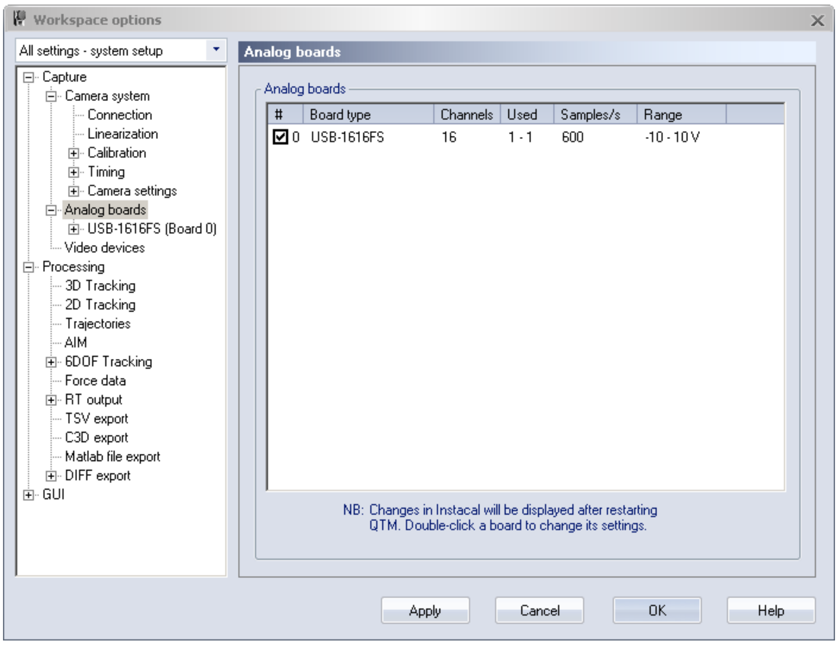
Note: Capture with MacReflex is no longer tested in QTM. The last version where it was thoroughly tested is QTM 2.0.387. However the files can be opened and edited in later version of QTM.



Analog boards

The analog boards that are installed in the measurement computer are listed on the **Analog boards** page, for information about analog boards see chapter “How to use analog boards” on page 356. For instructions on how to install an analog board, see “Analog capture hardware” on page E - 1.

Note: If more than one board is installed make sure that the synchronization cable is connected to all of the analog boards.



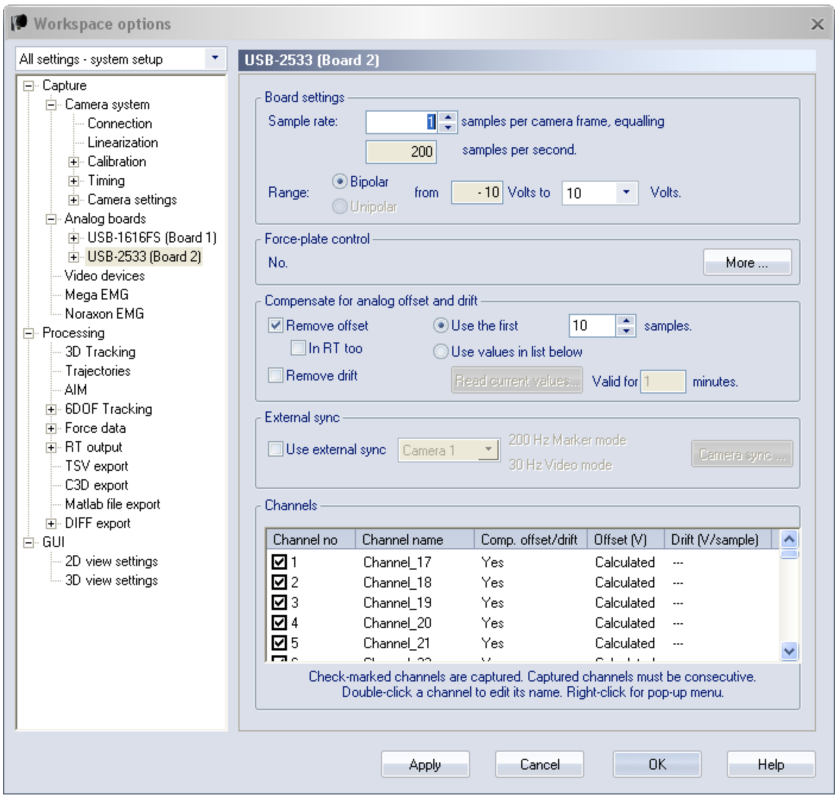
The list of the analog boards contains six columns: #, **Board type**, **Channels**, **Used**, **Samples/s** and **Range**. To use an analog board select the check box in the # column. Deselect the analog board to turn off the capture of analog data.

The **Board type** column shows the type of the analog board, for a list of boards that are compatible with QTM see “Analog capture hardware” on page E - 1.

The four last columns contain the settings for the analog board. To configure an analog board select the board on the list. Then the analog board appears below the **Analog boards** branch of the options tree. Click the name to go to the page with settings for the analog board, see chapter “Analog board (...)” on next page.

Analog board (...)

The page of each analog board contains the following settings: **Board settings**, **Force-plate control**, **Compensate for analog offset and drift**, **External sync**, **Channels**. For information on the settings see respective chapter below.



Board settings

Under the **Board settings** heading on the page for the analog board, there are settings for the sample rate and range of the analog board. When using this setting the analog board is only triggered by the camera system, for USB-2533 and USB-1616 each analog sample can be triggered by the camera, see chapter “External sync” on page 164.

Use the **Sample rate** option to select the samples per camera frame used for the analog capture. The default setting is one (1) sample per camera frame, i.e. the same sample rate as for the camera system. The actual sample rate for the analog board, in samples per second, is shown in the second text box. The sample rate can be different on separate boards, except for USB-1616FS where the sample rate is controlled by the master board.

Note: The minimum analog sample rate is 20 Hz, which is limited by the analog driver.

Note: When **External sync** is active the **Sample rate** option is disabled.

Use the **Range** options to select **Bipolar** range (both negative and positive voltage) or **Unipolar** range (only positive voltage) and to select the voltage range that is used for the analog capture. The default settings are **Bipolar** and ± **10 Volts**.

Force plate control

Under the **Force plate control** heading there is information about the setup of the control of the force plate. Click **More** to go to the **Force plate control settings** page.

Compensate for analog offset and drift

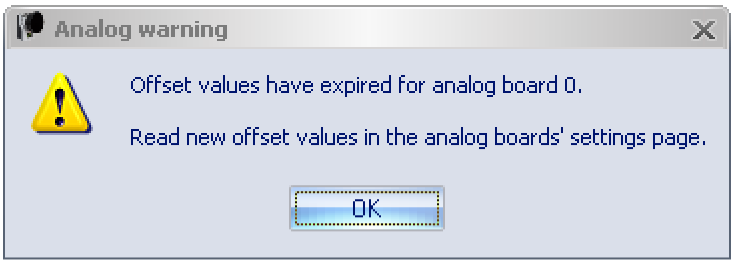
Under the **Compensate for analog offset and drift** heading there are settings for a filter which removes offset and drift. Activate the filter with the **Remove offset** and **Remove drift** options. The current compensation is displayed in the **Channels** list, see chapter “Channels” on page 165.

Remove offset
Removes an offset from the analog data. This setting can be used to remove the voltage level of a unloaded force plate. There are two options for the offset compensation and both are applied in RT/preview and in a file. However if none of the options are selected you can still turn on the compensation in a file from the **Data info** window, the compensation will then be calculated with the **Use the first and last** option.

Use the first
The offset is defined as an average of the samples in the beginning of the measurement. The number of samples used are defined in the text box next to the option.
Note: For this offset setting it is important that the samples in the beginning of RT/preview and a in file do not include any valid analog data. E.g. do not stand on the force plate at the start of the measurement since this data will be equal to 0 N if **Remove offset** is activated. In the case the analog signal is higher than 5 V QTM will give you a warning, see chapter “Analog offset warning” on page 357. However you can always remove the offset compensation manually in the file from the **Data info** window, see chapter “Analog data information” on page 82.

In RT too
Activate the **In RT too** option to remove the analog offset during RT/preview as well. Only activate this if you are using the RT data. For example if you zero a force plate amplifier after you started the RT/preview then the data will be wrong, because then the offset was calculated on the signal from the unzeroed amplifier. The data will however be OK when you capture the file, because then QTM calculates a new offset in the beginning of the file.

Use values in list below
The offset is saved before the measurement and displayed in the **Channels** list below. This option is not available in combination with **Remove drift**. Acquire the offset during RT/preview with the **Read current values** button. Make sure that the analog signal is correct when you press **Start** in the **Acquire offsets** dialog. Because the offset is then stored the analog signal can include valid data in the beginning of RT/preview and in a file. E.g. you can stand on the force plate when the measurement starts.
The offset is valid for the number of minutes specified next to the option. How long time depends on the drift of the analog signal. The analog warning below is displayed when you start a measurement and the time has expired.



Remove drift
Calculates a slope from the first and the last samples of the measurement. The number of samples used to calculate how much to remove from the analog data is defined in the text box next to the **Use the first and last** options. This slope is then deleted from the analog data. This setting can be used to remove a output drift of a force plate that slowly changes the voltage level during the measurement.
The drift compensation is only applied on a file.
Note: When the drift filter is activated it is important that the samples in the beginning and end of a in file do not include any valid analog data. E.g. do not stand on the force plate at the start and end of the measurement since this data is used to calculate the drift. However you can always remove the drift compensation manually in the file from the **Data info** window, see chapter "Analog data information" on page 82.

External sync

The **Use external sync** option can be used to frame synchronize the analog capture with sync signal from the camera system. This option is only available for the analog boards USB-2533 and USB-1616FS. For instructions on how to connect the sync signal from the camera to the analog board see chapter "Connection of analog board" on page 356.

When activated each sample on the analog board is triggered by a TTL pulse from the camera system, which means that there is no drift between the two systems. In an Oqus system you can use any frequency that is a multiple or divisor of the camera frequency. However on a ProReflex system you can only use the same frequency as the capture frequency, which means that the only available option for ProReflex is **Use external sync**.

The following settings are available for the external sync.

Camera selection (Oqus)
Select the camera where the Control port splitter is connected. The splitter can be connected to any camera in the system but you must specify which it is, because the sync out signal can be changed on an individual camera.

Analog frequency
These are the sync out frequencies of the selected which will be the analog frequency. The frequency is shown both for **Marker mode** and **Video mode**, because the marker capture rate and video capture rate can differ. For ProReflex only the analog frequency in marker mode is shown because it cannot capture video. The frequency can be different on separate boards, except for USB-1616FS where the sample rate is controlled by the master board.

Note: The minimum analog sample rate is 20 Hz, which is limited by the analog driver.

Note: When **Use external sync** is activated the **Sample rate** option on the top of the page is disabled.

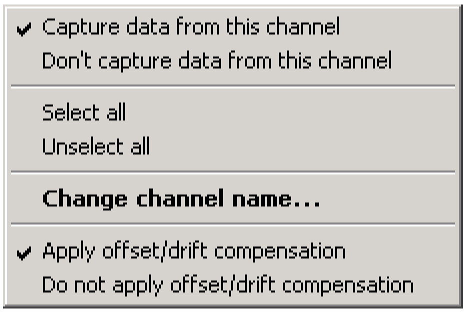
Note: If the Reduced real-time frequency is used during preview, then the analog frequency during preview may differ from the ones reported on this page. The ratio between the camera frequency and the analog frequency set on the **Advanced (Timing)** page will however be the same, so you can calculate the analog frequency.

Camera sync (Oqus)
To change the sync output frequency click on **Camera sync** to go directly to the **Timing** settings for the selected camera. There you can use any option for

Synchronization output that you like, see chapter “Synchronization output” on page 137. The recommended option in most cases is **Camera frequency multiplier** because then the analog frequency will be a multiple of the camera frequency. This is for example a requirement for the C3D file.

Channels

The channels of the analog boards are listed under the **Channels** heading. To use a channel select the corresponding check box in the **Channel no** column. The selected analog channels must be in successive order. For example, if only channel no 3 and channel no 7 are used all channels in between must also be selected. Double-click on the a row to change the **Channel name** and right-click for the menu below.



Capture data from this channel
Use this channel, the same as selecting the checkbox.

Don't capture data from this channel
Do not use this channel, the same as deselecting the checkbox.

Select all
Use all of the channels

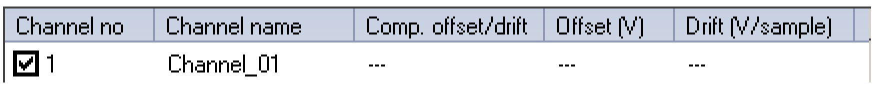
Unselect all
Use none of the channels. Sometimes the fastest way to select a new range of channels.

Change channel name...
Open a dialog to change the channel name.

Apply offset/drift compensation
Use this options to apply offset and/or drift compensation. However you can always turn off the compensation in a file from the **Data info** window. This setting applies to all channels that are selected (i.e. the line is blue).

Do not apply offset/drift compensation
Use this option to not apply the offset and/or drift compensation. However you can always turn on the compensation in a file from the **Data info** window. This setting applies to all channels that are selected (i.e. the line is blue).

The current settings under the **Compensate for analog offset and drift** heading is displayed in the last three columns. The different settings are described below.



If no compensation is activated all three columns display ---. This means that no compensations is applied in RT/preview or in a file. However you can turn on the compensation in a file from the **Data info** window. The compensation will be calculated from the **Use the first and last** option.

Channel no	Channel name	Comp. offset/drift	Offset (V)	Drift (V/sample)
☑ 1	Channel_01	Yes	Calculated	Calculated

If the compensation is activated with the option **Use the first and last**, then the **Offset** and **Drift** columns says **Calculated**. The offset compensation will be applied in both RT/preview and in a file and calculated from the first samples, which means that it is important that there is no signal on the analog channel and the start of RT/preview or in the beginning of a file.
The drift compensation is however just applied in the file, since it isn't calculated for the RT/preview. For the drift compensation it is important that there is no analog signal in the beginning and the end of the file.

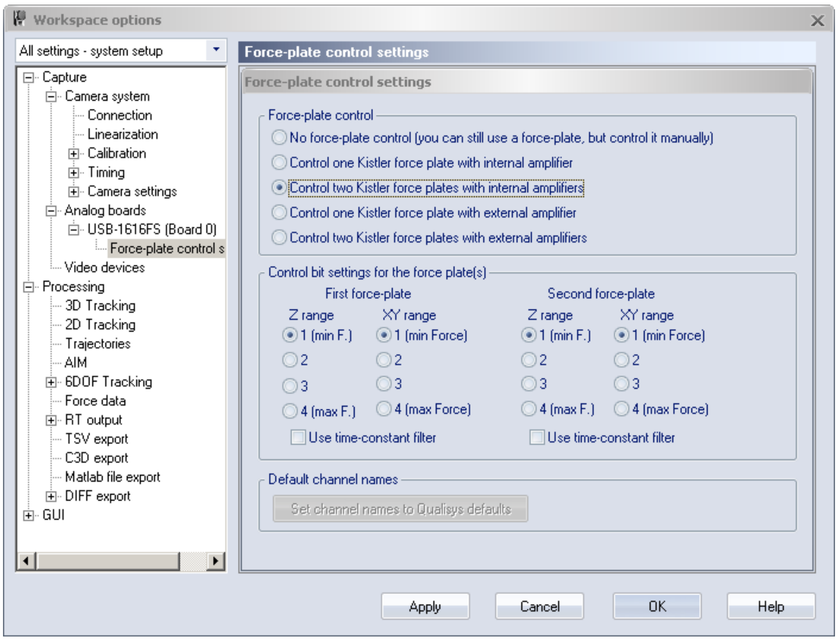
Channel no	Channel name	Comp. offset/drift	Offset (V)	Drift (V/sample)
☑ 1	Channel_01	Yes	-0.441	---

If the compensation is activated with the option **Use values in list below**, then the last read value is displayed in the **Offset** column. The offset compensation will be applied in both RT/preview and in a file. However since the offset values have already been stored there can be an analog signal in the beginning of RT/preview and a file. Start RT/preview and use the **Read current values** button to updated to offset values for the channels where the compensation is activated.
The **Drift** column displays ---, because the drift cannot be compensated for in this mode.

Channel no	Channel name	Comp. offset/drift	Offset (V)	Drift (V/sample)
☑ 1	Channel_01	No	---	---

When the compensation of a channel is deactivated by the **Do not apply offset/drift compensation** option, then the **Comp. offset/drift** column displays No. The compensation is then not applied, but it can be activated in a file from the **Data info** window.

Force plate control settings



The **Force plate control settings** page contains settings for the control of the force plate.

Note: These settings are only valid for the Kistler force plates. For another type of force plate the **No force plate control** option should be used. The Kistler force plates can, however, be externally controlled regarding Operate/Reset and charge amplifier range settings.

Note: If you have used a previous installation with the QTrac software and a Kistler force plate with internal (built-in) amplifier, it uses a signal converter to reverse the control signals to the force plate. When using the QTM software this converter must be removed.

Force plate control

Under the **Force plate control** heading there is a list of possible combinations of Kistler force plates. Select the correct configuration of Kistler force plates that are used in the analog capture. If you do set this option there is no operate/reset signal sent to the Kistler plate and it must then be controlled manually on the Kistler amplifier or Connection box.

The following configuration options are available:

- No force plate control. This option is used when no Kistler force plate is connected to the analog acquisition system or if it is controlled manually.
- Control one Kistler force plate with internal amplifier
- Control two Kistler force plates with internal amplifier
- Control one Kistler force plate with external amplifier
- Control two Kistler force plates with external amplifier

Control bit settings for the force plate(s)

Under the **Control bit settings for the force plate(s)** heading there are options for controlling the range of the amplifier and the use of time-constant filter of the Kistler force plate.

The range of the amplifier can be different for the vertical (Z range) and horizontal (XY range) force components. The selected range determines the magnitude of the forces that can be measured with the force plate.

The **Time-constant** filter is useful for long measurements of static forces, for regular measurements the **Time-constant** filter should not be used.

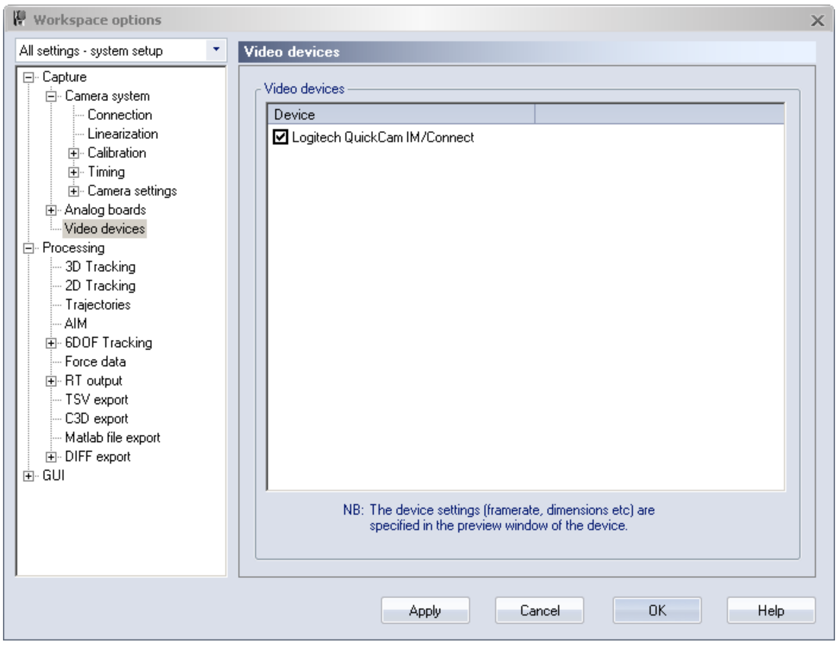
For further information about the range settings and the time-constant filter, see the manual for the Kistler force plate.

Default channel names

Under the **Default channel names** heading the default names of the Kistler force plate's analog signals can be applied to the names of the analog channels on the analog board.

Note: *This function is not yet implemented.*

Video devices



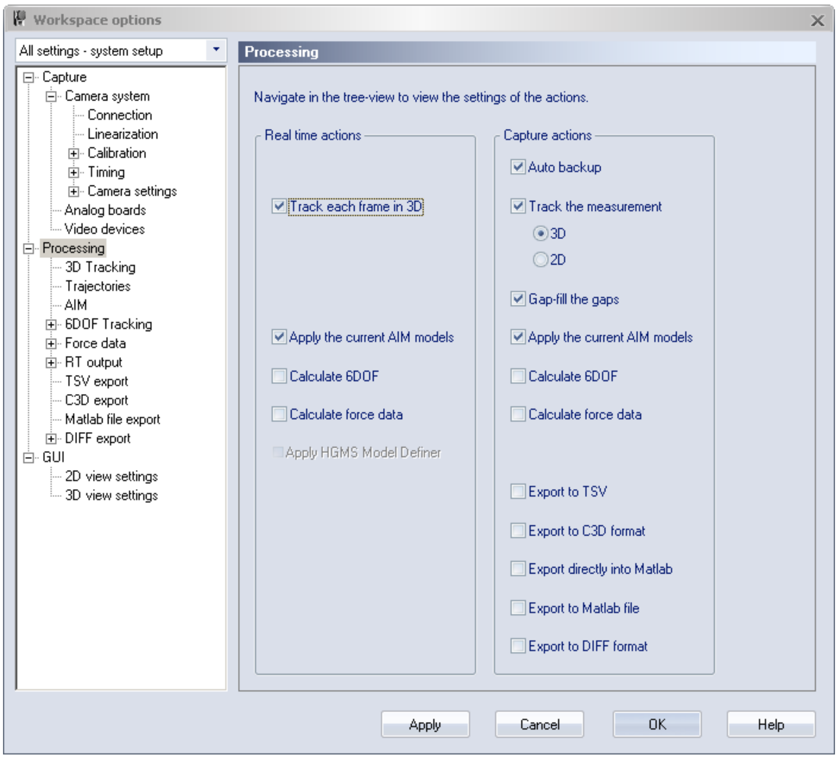
The external video devices that are installed in the measurement computer are listed on the **Video devices** page. This include DV/webcam and Point Grey cameras, for more information on how to use them see chapter “DV/webcam video in 2D view” on page 42 and “How to use Point Grey cameras” on page 367.

To use a video device select the check box next to it and then the video will be displayed in the 2D view window. To open a 2D view window with just the video cameras you can click on the **Video** button in the toolbar.

Note: If you run QTM in **Immediate** mode, the rate of the video capture might be slightly reduced. However, to improve the rate QTM could be set to **Frame buffering** or **Segment buffering** mode.

Note: QTM supports simultaneous video recording from several video devices, but there can be some problems with the capture of video data if there are too many cameras. E.g. the frame rate of the video devices may be lowered and sometimes the video is not captured at all, it depends a lot on how fast the computer is.

Processing



The **Processing** branch of the options tree contains options for actions that can be performed on the 2D data. The options have an effect on the 2D data in real-time (preview), during a measurement and after the measurement, for information see chapters “Processing measurement data” on page 277 and “Real time mode” on page 271. For batch process a similar processing page is used see chapter “Batch processing” on page 279.

The actions that can be performed are as follows:

- **Auto backup** only Capture action, see chapter “Auto backup” on page 249.
- **Track each frame**
- **3D**
- **2D** only Capture action
- **Gap-fill the gaps** only Capture action
- **Apply the current AIM model**
- **Calculate 6DOF**
- **Calculate force data**
- **Apply HGMS Model Definer** only Real time action. For more information see the HGMS manual.
- **Export to TSV** only Capture action
- **Export to C3D format** only Capture action
- **Export directly into Matlab** only Capture action

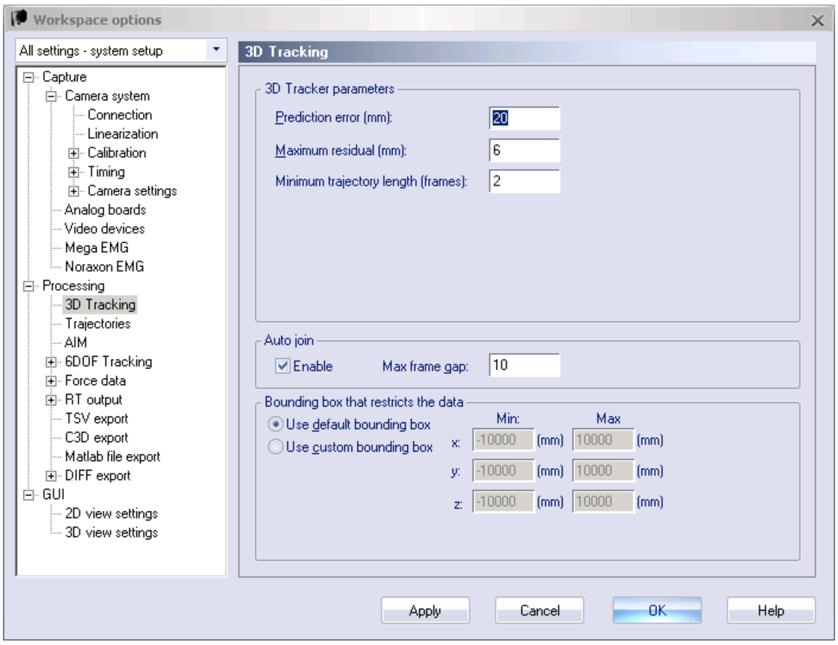
- **Export to Matlab file** only Capture action
- **Export to DIFF format only Capture action** only capture action

The actions can be defined separately for Real-time and Capture. Select each action that will be used from the processing steps list. The settings for each option can be reached by clicking on the corresponding page in the tree to the left.

⚠ **Important**: 3D and 2D tracking are mutually exclusive and can therefore not be used at the same time.

3D Tracking

The **Tracking** page contains settings for the 3D tracking of the motion capture data. The 3D tracker uses 2D data of the cameras in the system to calculate a trajectories in a 3D view, see chapter “3D tracking measurements” on page 283.

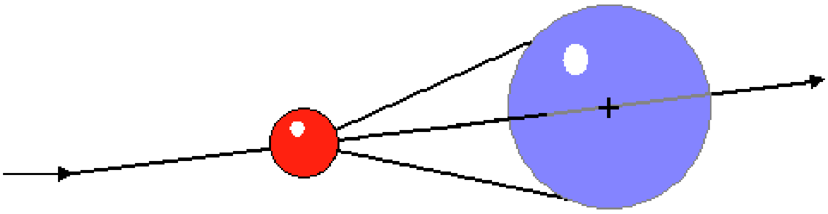


3D Tracker parameters

Under the **3D Tracker parameters** heading there are three parameters for the 3d tracker:**Prediction error**,**Maximum residual** and **Minimum trajectory length**.

The 3D tracking function uses a small buffer (4 - 10 frames) of previous positions of a trajectory to predict the next location of the trajectory’s marker in 3D space. The function uses a very general algorithm with just a few tracking parameters that adjust the equations to specific tracking problems.

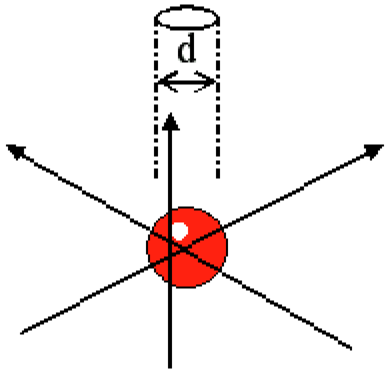
Prediction error



The **Prediction error** parameter specifies the maximum distance (in mm) between a predicted position of a trajectory and a captured point that is allowed for it to be assigned to that trajectory. The parameter therefore provides a margin of error with which a 3D point may deviate from the mathematically calculated next position. Since real-world data cannot be expected to exactly fit an equation, this provides a mechanism for dealing with the unpredictability of the real changes in a marker’s trajectory. The example above shows the 3D point (the red ball) and its predicted next position (the black cross). The blue sphere is the volume in which the next 3D point is accepted as part of this trajectory.

The default value of the **Prediction error** is 30 mm. If the **Prediction error** is larger than half of the closest distance between two markers in the measurement, there is a possibility of trajectory cross-over or swapping between the two trajectories. Therefore, if erratic or jumpy motions within single trajectories are seen, then it is likely that the parameter has been set too high. On the other side if the value is set too small, excessive division of trajectories (high segmentation) will occur, resulting in many more trajectories than markers. The reason is that the distances because of noise and acceleration might be greater than that allowed for by the **Prediction error** parameter. Therefore, if high segmentation or gaps in the data are experienced it is likely that the parameter is set too low.

Maximum residual



The 3D tracking function uses the **Maximum residual** parameter together with the **Prediction error** to control the continuation of already started trajectories. The **Maximum residual** sets a limit (in mm) to the distance from the final location of the 3D point within which all intersecting 2D marker rays are considered to belong to that 3D point. The example above shows three 2D marker rays where the intersections are not in the same point, but they are all within the **Maximum residual** (d) of the 3D point (the red ball). The parameter is shown as a circle but in reality it is of course a sphere around the 3D point.

The default value of the **Maximum residual** is 10 mm. The value for the parameter can usually be set to 2 - 5 times the average residual values in the calibration result for the camera system. A too large value will slow down the calculation and will tend to produce merged 3D points, resulting in defective trajectories. On the other hand a too small value will probably cause ghost markers and high segmentation, resulting in many more trajectories than markers.

Minimum trajectory length

The **Minimum trajectory length** parameter defines the minimum number of frames for a trajectory. The default value is 2 frames, which is also the minimum value since a trajectory that is only 1 frame long is not really reliable.

Increase the number of frames if you have a lot of short extra trajectories. When those are removed the AIM process will also work better. However it is a good idea to check so the extra trajectories are not caused by something else, e.g. a bad calibration or something reflective.

Auto join

As soon as a marker is completely hidden during the measurement, even if it is just for one frame, a new trajectory will be started by the tracking function. The **Auto join** function examines the trajectories after the tracking to see if any can automatically be joined. The **Max frame gap** specifies the number of frames allowed between two trajectories that will be qualified for joining. The **Auto join** function then uses the tracking parameters to decide if two trajectories can be joined.

Note: The **Auto join** function is enabled by default with a value of 10 frames, and it should be used to make the identification easier.

Bounding box that restricts the data

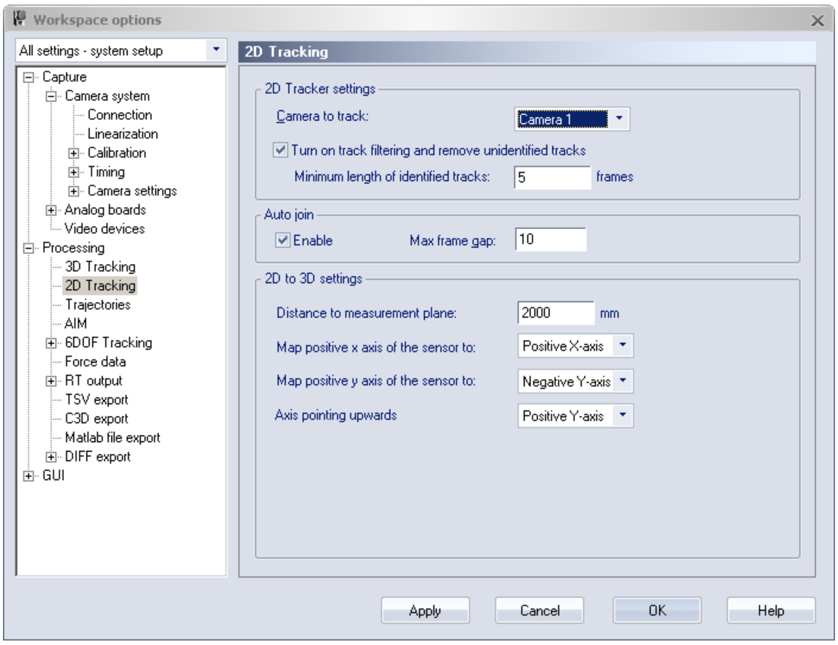
The bounding box is the volume in which the trajectories will be calculated. It can be helpful in removing unnecessary or unwanted markers from the final motion capture measurement, by deleting the volume that is unimportant for the measurement. The bounding box is displayed as a white outlined box in the 3D view, the display is controlled on the **3D view settings** page.

By default the **Use default bounding box** option is selected, which means that the bounding box is a cube with a side of 20 m with the center in origin of the coordinate system of the motion capture.

With the **Use custom bounding box** option, a box can be specified that limits the volume of the measurement. I.e. only 3D points within this bounding box will be calculated. The box is specified as the distances of each direction (in mm) to the origin of the coordinate system of the motion capture.

Note: If you use a custom bounding box and then translate or rotate the global coordinate system the bounding box will also be moved. This may result in that you move the bounding box outside the wanted measurement volume. Therefore the bounding box must be corrected manually after the transformation.

2D tracking



The **2D tracking** page contains settings for the 2D tracker. The 2D tracker uses the data of just one camera to calculate trajectories in a plane in the **3D view** window, see chapter “2D tracking of data” on page 298.

Tracking settings

Under the **Tracking settings** heading you can select the camera that will be tracked with the **Camera to track** option. Choose the camera from the drop-down list. You can only 2D track one camera at a time.

The option **Turn on track filtering and remove unidentified tracks** is very useful to limit the number of trajectories in the 2D tracking output. It filters the data so that short identified tracks and the 2D markers that have not been identified are not displayed as trajectories. Use the option **Minimum length of identified tracks** to set the minimum number of frames for an identified track in the 2D data to be transferred to a 3D trajectory.

Important: If you turn off the filter the unidentified markers in the 2D data will be transferred to a 1 frame long trajectory. This means that the data will be very fragmented if the 2D tracker has left a lot of markers unidentified.

Auto join

As soon as a marker is completely hidden during the measurement, even if it is just for one frame, a new trajectory will be started by the tracking function. The **Auto join** function examines the trajectories after the tracking to see if any can automatically be joined. The **Max frame gap** specifies the number of frames allowed between two trajectories that will be qualified for joining. The **Auto join** function then uses the tracking parameters to decide if two trajectories can be joined.

Note: The **Auto join** function is enabled by default with a value of 10 frames, and it should be used to make the identification easier.

2D to 3D settings

The **2D to 3D settings** heading contains settings for how the 2D data is displayed in **3D view** window.

Distance to measurement plane
Set the distance from the camera sensor to the measurement plane in mm. The measurement plane will be the same as the green grid in the **3D view** window. With this setting you can get the distances in the **3D view** window to match the real distances. You must measure the distance yourself, but for most applications the accuracy of the distance does not have to be better than in cm.

Map positive x axis of the sensor to

Select the axis that will map to the positive x axis of the sensor, i.e. to the right if looking in the direction of the camera. Choose the axis from the drop-down list.

Map positive y axis of the sensor to

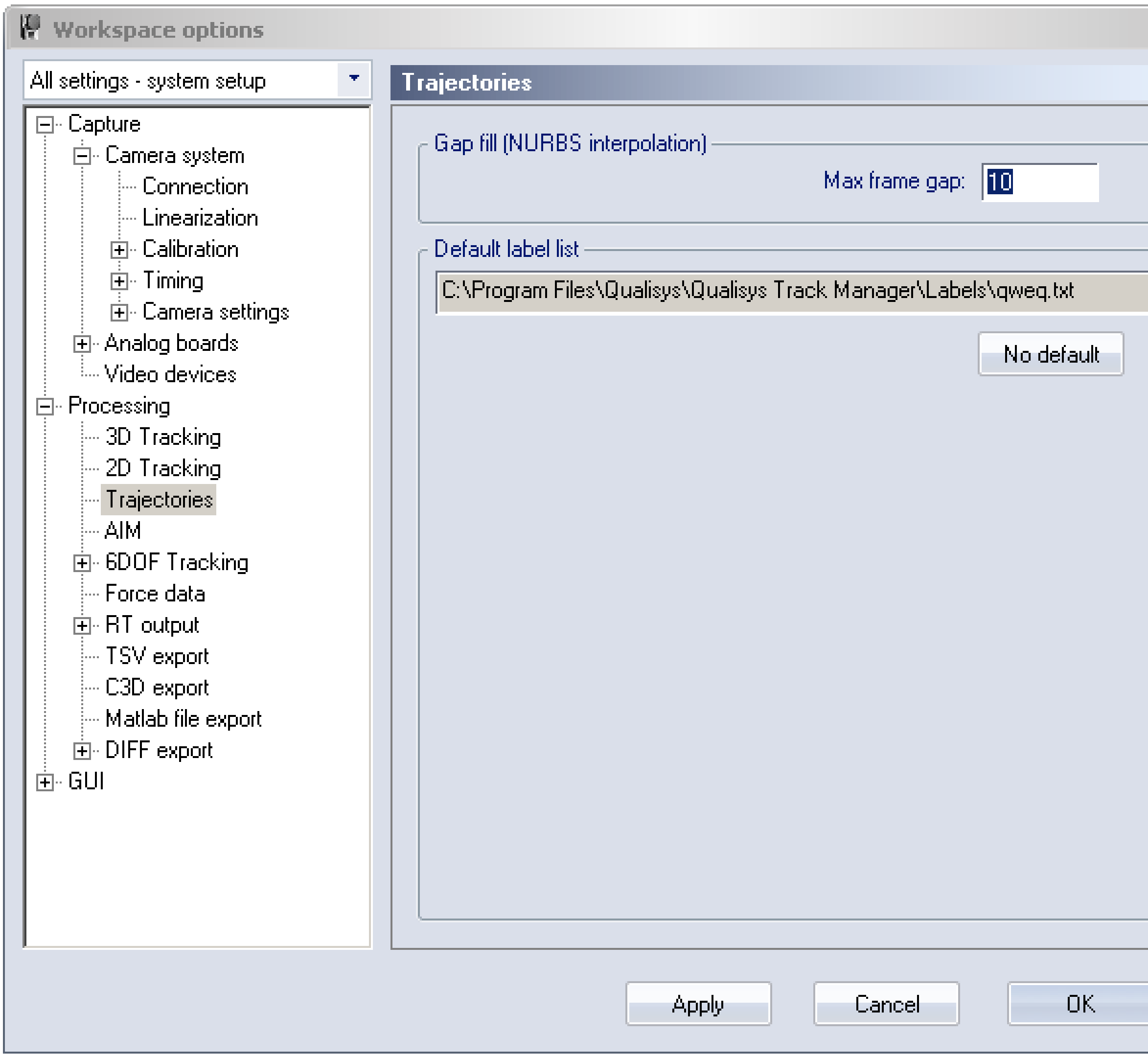
Select the axis that will map to the positive y axis of the sensor, i.e. downwards in the camera view. Choose the axis from the drop-down list.

Axis pointing upwards

Select the axis that will be pointing upwards. Choose the axis from the drop-down list. With this setting you can move the camera in the 3D view so that it looks from different directions, i.e. from the side, from above or from below.

Trajectories

The **Trajectories** page contains two settings:**Gap fill (NURBS Interpolation)** and **Default label list**.



Gap fill (NURBS interpolation)

Under the **Gap fill (NURBS Interpolation)** heading there is a **Max frame gap** option. The trajectories will only be gap filled if the gap between the two parts is less than the **Max frame gap**.

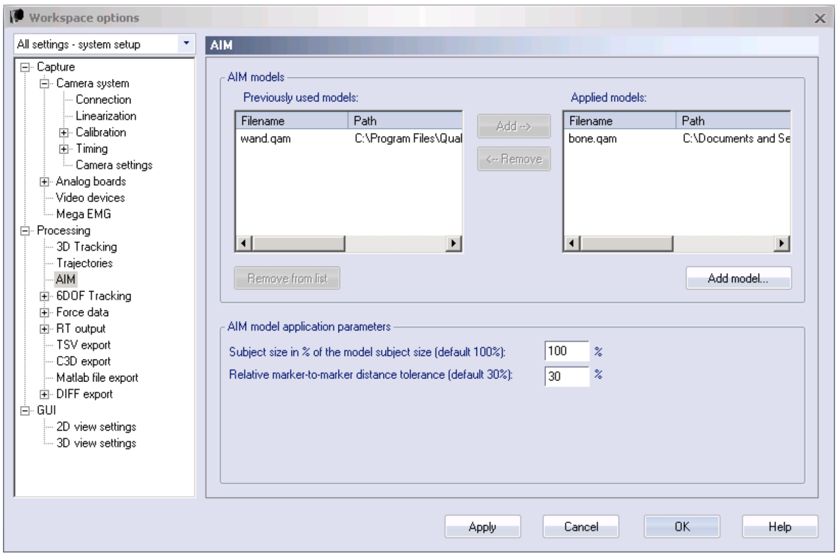
The gap fill function is applied to trajectories either via the **Trajectory info window** menu or as a processing step. If gap fill is applied from the **Trajectory info window** menu, it is only the selected trajectories that are gap filled. When gap fill is performed as a processing step gap fill is tried on every trajectory.

Default label list

Under the **Default label list** heading, a label list with empty trajectories can be specified. It will be loaded automatically in the **Labeled trajectories** window after each measurement. Click **Browse** to select the label list. Click **No default** to remove the default label list.

AIM

The **AIM** page contains settings for the Automatic Identification of Markers (AIM) function, for information about AIM see chapter "Automatic Identification of Markers (AIM)" on page 302.



AIM models

Under the **AIM models** heading there are two lists with AIM models. The **Applied models** list are the models that are used by QTM. There can be several models in this list and then QTM will try to apply them all. Remember to **Remove** the unnecessary models from this list otherwise the AIM application might fail. Add a saved model to the list with **Add model**.

The **Previously used models** list contains all the models that have been used by QTM. If you want to use a model again click on **Add** to move the model to the **Applied models** list. Remove a model from the list by clicking on **Remove from list**, it will only remove the model form the list the AIM model file is not deleted.

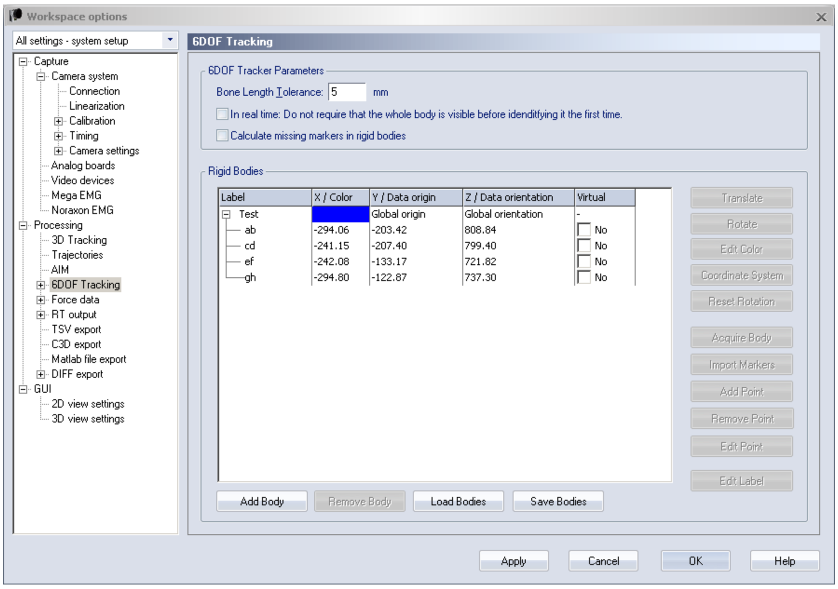
AIM model application parameters

Under the **AIM model application parameters** heading you can set the following settings that adjust the application of the AIM model.

Subject size in % of the model subject size
Adjust the size of the model to fit the measurement object, the setting is in percent of the original model subject. E.g. you can dimension the model of an adult to fit a child by reducing the size.

Relative marker to marker distance tolerance
Change the permitted marker to marker distance relative the model marker to marker distance. As default any marker that is within ±30 % of the model's marker to marker distance will be tried in the model application.

6DOF tracking



The **6DOF Tracking** page contains the **6DOF Tracker parameters** and the list of **Rigid bodies**. The 6DOF tracker uses this information to calculate the position and rotation from the 3D data, see chapter “6DOF tracking identifies rigid bodies from the 3D data of a measurement and calculates their translation and rotation with respects to the global coordinate system. It is activated with Calculate 6DOF on the Processing page in the Project options dialog and is controlled by the 6DOF tracking parameters, see chapter “6DOF tracking” on page 179.” on page 285. For description of the **Rigid bodies** list see chapter “Rigid bodies” on next page.

It is also possible to export the 6DOF tracking output to another computer and as an analog signal, see chapter “RT output” on page 203 respectively chapter “6DOF analog export” on page 205.

6DOF Tracker parameters

Specify the tracking parameter for the 6DOF tracking under the **6DOF Tracker parameters** heading. The settings are the **Bone length tolerance** and an option for real time tracking.

Bone length tolerance
The **Bone length tolerance** (in mm) is the maximum separation between the lengths of the corresponding bones in a rigid body definition and a measured rigid body. E.g. if the **Bone length tolerance** is specified to 5.0 mm and the current bone in the rigid body definition is 100.0 mm, then the measured separation between two markers must be in the range of 95.0 - 105.0 mm for the tracker to accept the possibility that the calculated markers may be the pair specified for the rigid body.

The default value of the **Bone length tolerance** is 5 mm. Increase the value of the parameter if the 6DOF tracking cannot find the body. Decrease the value of the parameter if a body is found but the orientation or something else is wrong.

The effect of the **Bone length tolerance** differs slightly between RT and in files. In RT the marker that is outside the tolerance will be unidentified and the 6DOF body will be calculated from the remaining markers. In a file the automatic 6DOF tracker will discard the whole trajectory that is wrong and then calculate the 6DOF body from the other trajectories. However if you identify the trajectories manually and then just **Calculate 6DOF**, then there will be no 6DOF data in frames where a marker is outside the tolerance.

In real time
Enable the '**In real time**' option to remove the requirement that all markers must be to identify the body. This can help the real time tracker if it is hard to see all of the markers.

Calculate missing markers in rigid bodies
Active the option to calculate virtual trajectories for lost 6DOF markers in RT and files, for more information see chapter “Virtual markers calculated from 6DOF data” on page 292.

For information on 6DOF tracking see chapter “6DOF tracking identifies rigid bodies from the 3D data of a measurement and calculates their translation and rotation with respects to the global coordinate system. It is activated with Calculate 6DOF on the Processing page in the Project options dialog and is controlled by the 6DOF tracking parameters, see chapter “6DOF tracking” on page 179.” on page 285.

Rigid bodies

The **Rigid bodies** list contains the definition of the 6DOF bodies. The bodies are used by the 6DOF tracking to find the measured rigid bodies in a motion capture. The list consists of five columns: **Label**, **X / Color**, **Y / Data origin**, **Z / Data orientation** and **Virtual**.

Label
The **Label** column contains the name of the rigid body and its points. Double-click on the name of the rigid body or the points to edit them. The points can have any name, however if the same name is used in another 6DOF body or an AIM model then you need to follow the instructions in chapter “”.

Color
The color of the rigid body is displayed in the **X / Color** column on the same row as the name of the rigid body. Double-click on the color to open the **Color** dialog where any color can be selected. The color is used in the **3D view** window for the markers of the rigid body and for its name.
Note: The 3D trajectories automatically have a slightly brighter color than the 6DOF body and therefore it will look like the markers have two colors.

Data origin and **Data orientation**.
The coordinate system that the 6DOF body data refer to is displayed in the **Y / Data origin** and **Z / Data orientation** columns on the same row as the name of the rigid body. Double-click on either the origin or the orientation setting to open the **Coordinate system for rigid body data** dialog, see chapter “Coordinate system for rigid body data” on page 184.

X, **Y**, **Z**
The **X**, **Y** and **Z** columns contain the coordinates of the points in reference to the local origin. Double-click on the coordinates of to edit them.

Virtual
Select this option to make a point in the 6DOF body virtual, see chapter “Virtual markers calculated from 6DOF data” on page 292.

The options that are used to edit the list are described below:

Add body
Add a new body to the **Rigid bodies** list. The new body will be empty and called 'New Body #1', 'New Body #2' and so on.

Remove body
Remove the selected body from the **Rigid bodies** list.

Load bodies
Loads bodies to the **Rigid bodies** list from an XML file or from a BOD file. The BOD file is an old format for rigid body definitions

Note: **Load bodies** will overwrite any other bodies in the list.

Save bodies
Save the bodies in the **Rigid bodies** list to an XML file. Name the file and click **Save**, the file is saved in the same folder as the Label list files. The file can be edited in a text editor, e.g. Notepad.

Note: Make sure that all of the bodies for the measurement are in the same file, since **Load bodies** overwrites the bodies in the list.

Translate
With **Translate** the local origin of the 6DOF body definition can be moved to any place in reference to the points of the body, which means that the rotation center of the body is changed. The local origin is also the origin of the coordinate system that represents the 6DOF body in the 3D view. Click **Translate** to open the **Translate body** dialog, see chapter "Translate body" on next page.

Rotate
With **Rotate** the pitch, roll and yaw of the local coordinate system is changed. This will change the orientation of the local coordinate system in reference to the global coordinate system. I.e. it changes the rotation of the rigid body where its roll, pitch and yaw are zero in reference to the global coordinate system. Click **Rotate** to open the **Rotate body** dialog, see chapter "Rotate body" on page 183.

Edit color
Open the **Color** dialog where any color can be selected. The color is used in the **3D view** window for the markers of the rigid body and for its name.

Note: The 3D trajectories automatically have a slightly brighter color than the 6DOF body and therefore it will look like the markers have two colors.

Coordinate system
Change the definition of the local coordinate system, see chapter "Coordinate system for rigid body data" on page 184.

Reset rotation
This will reset the orientation of all the rigid bodies in the list. Reset means that the local coordinate systems will be aligned to the global coordinate system and all the angles will therefore be zeroed.

Note: This does not work if the reference has been changed in the **Coordinate system for rigid body data** dialog so that it is not the global coordinate system.

Acquire body
Acquire the rigid body definition from the current marker positions in RT/preview mode, see chapter "Acquire body" on page 185.

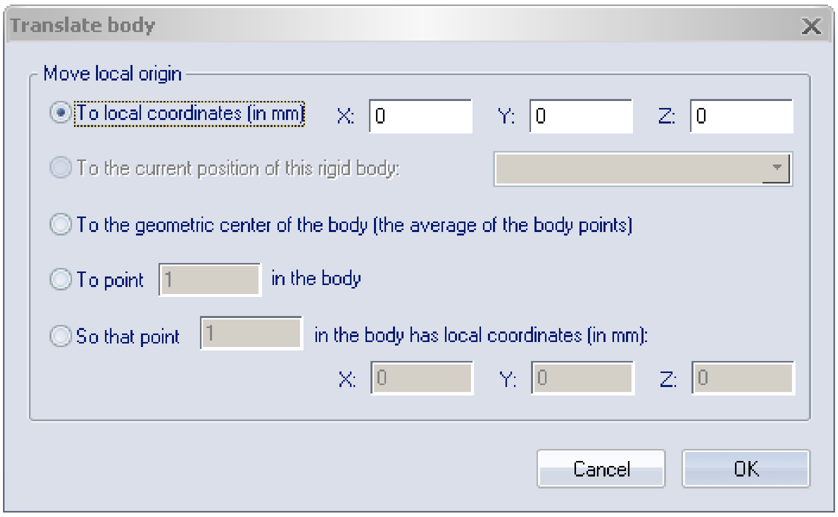
Import markers
Import marker positions from the currently open file to use them as in a rigid body definition, see chapter "Import markers" on page 185.

Add point
Add a point to the selected rigid body.

Remove point
Remove the selected point.

Edit point
Edit the selected point. Use Tab and Shift+Tab to go to the next respectively the previous coordinate.

Translate body



The **Translate body** dialog contains the following five ways to translate the local origin:

To local coordinates (in mm)
Specify the translation of the local origin in the X, Y and Z direction of the local coordinate system. E.g. if the local origin is translated 1 mm in the X direction all of the points' X coordinates will be 1 mm less than before.

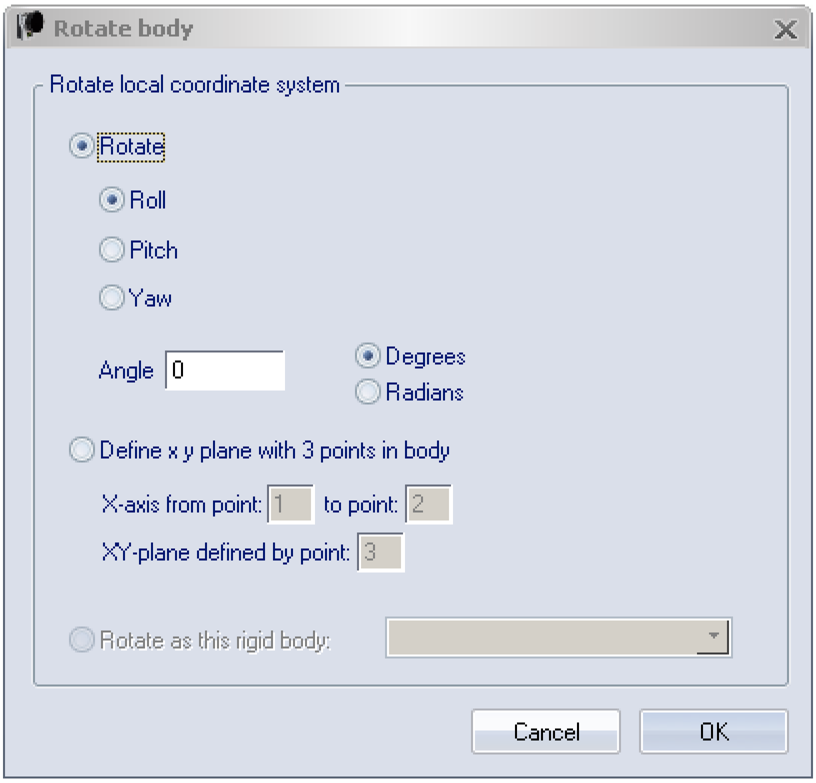
To the current position of this rigid body
Move the local origin to current position of the selected rigid body in the list. This will zero the position of the body in the list if its position is referred to the current rigid body.

To the geometric center of the body (the average of the body points)
Move the local origin to the geometric center of all the points in the rigid body definition. The geometric center can be seen as the center of mass of a body with the same weight at all of the points.

To point in the body
Move the local origin to one of the points in the rigid body definition. Enter the number of the point that is used as local origin.

So that point ... in the body has local coordinates (in mm)
Move the local origin so that one of the points in the rigid body definition has a desired position. Enter the number of the point and the position in X, Y and Z direction (local coordinate system).

Rotate body



The **Rotate body** dialog contains the following two ways to rotate the local coordinate system:

Rotate
Rotate the local coordinate system clockwise around one of the axes, when looking in the positive direction.

Roll
The axis of rotation is the X-axis.

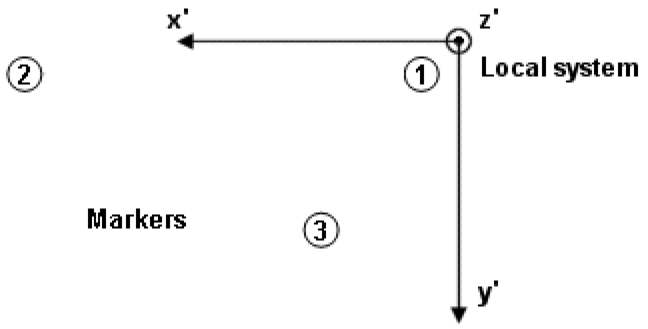
Pitch
The axis of rotation is the Y-axis.

Yaw
The axis of rotation is the Z-axis.

Angle
Choose the angle of rotation either in **Degrees** or in **Radians**.

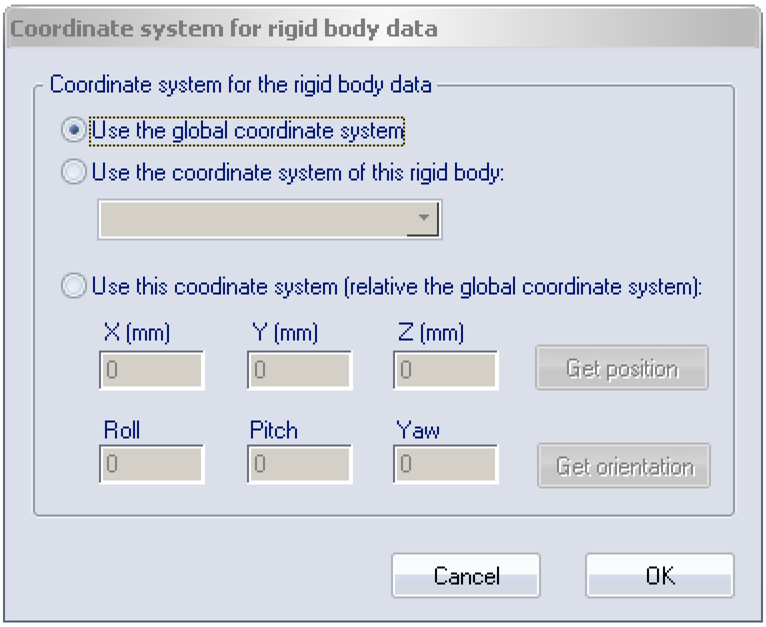
Note: The description above is for the **Qualisys standard** for the Euler angles. If you have changed the definition on the **Euler angles** page the new settings will be used in this dialog.

Define x y plane with 3 points in body
Define the rotation of the local coordinate system with three points from the rigid body definition. The x axis is defined by the points in the **X-axis from** setting. The x-axis direction will be from the first point to the second. Then xy-plane is defined by the point in the **XY-plane defined by** setting, see image below.



Rotate as this rigid body
Rotate the local coordinate system to current orientation of the selected rigid body in the list. This will zero the orientation of the body in the list if its orientation is referred to the current rigid body.

Coordinate system for rigid body data



To describe the position and orientation of the 6DOF body its data must be referred to another coordinate system. By default the data is referred to the position and orientation of the global coordinate system. However, with the settings on the **Coordinate system for rigid body data** dialog you can refer the local coordinate system to the following alternatives of coordinate system origin and orientation.

Origin of the coordinate system for the rigid body data

Use the global coordinate system
The position is referred to the origin and orientation of the global coordinate system.

Use the coordinate system of this rigid body
The position is referred to the current position and orientation of another rigid body. This means that if the reference body moves the 6DOF body data will change even if that body is stationary. Select the body from the drop-down list.

Note: Using another body as reference will increase the noise in the 6DOF data, especially if there is a long distance between the two bodies. This is because a small rotation error in the reference body will result in a much larger noise in rigid body.

Note: If the reference 6DOF body cannot be tracked the 6DOF body will disappear in the **Data info** window. However the 6DOF data is always saved and displayed in the **3D view** window so that if the file is reprocessed with another reference the 6DOF body will appear again.

Use this coordinate system (relative the global coordinate system)
The position is referred to a stationary point defined in the global coordinate system. Define the position in mm in the three directions (**X**, **Y** and **Z**) and orientation in degrees for **Roll**, **Pitch** and **Yaw**.
Note: Roll, pitch and yaw is the **Qualisys standard**, but if the Euler angles definition are changed on the **Euler angles** page the new settings will be used in this dialog.

With **Get position** and **Get orientation** the current position or orientation is acquired. Which means that the data will be zeroed for the current position of the rigid body.

Acquire body

With **Acquire body** a rigid body definition can be acquired from preview mode. Place the rigid body with the markers in the measurement volume and open a new file with **New** on the **File** menu. Open the **6DOF tracking** page in the **Project options** dialog and click **Acquire body** to open the **Acquire body** dialog.

Specify the number of frames to collect with the **Frames to acquire** setting. Click **Acquire** to start the acquisition. The points of the rigid body definition are then calculated from the average of each marker's position in these frames. The **Stop** option can be used to cancel the collection before all frames have been captured.

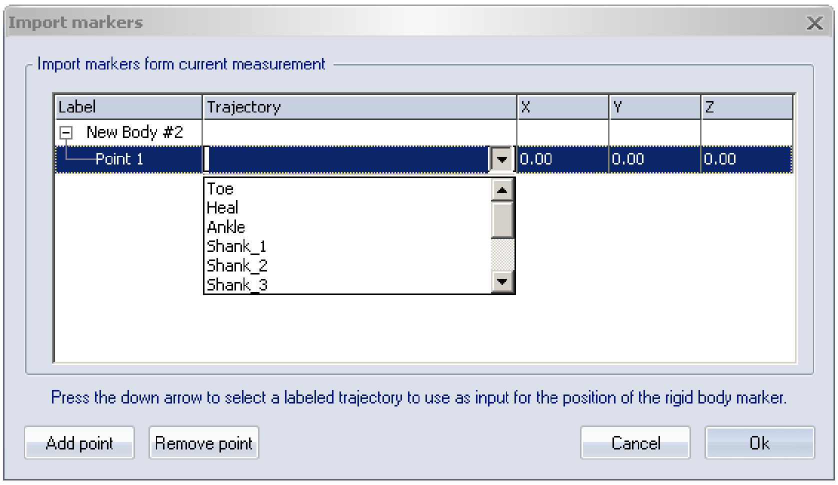
To see that the 6DOF tracking can find the body, change to 6DOF tracking on the **Processing** page and click **Apply**. The body should appear in the 3D view.

Note: The measurement must be done on a stationary rigid body with at least four markers and the body cannot be flat.

Note: It is a good idea to place the body so that the orientation of the desired local coordinate system is aligned with the global coordinate system. It is also a good idea to place the desired origin of the local coordinate system in the origin of the global coordinate system. Another way to easily define the local origin of the body is to use an extra marker placed at the desired location of the local origin. After acquiring the body coordinates, use the **Translate body** dialog to translate the local origin to the location of the extra marker. Then delete the extra marker from the body definition with **Remove point**.

Import markers

With **Import markers** the average position of a trajectory from the current measurement can be imported to a point in the rigid body definition. Since it is the average of all the frames in the measurement it is important that the trajectory is stationary. Click **Import markers** to open the **Import markers** dialog.



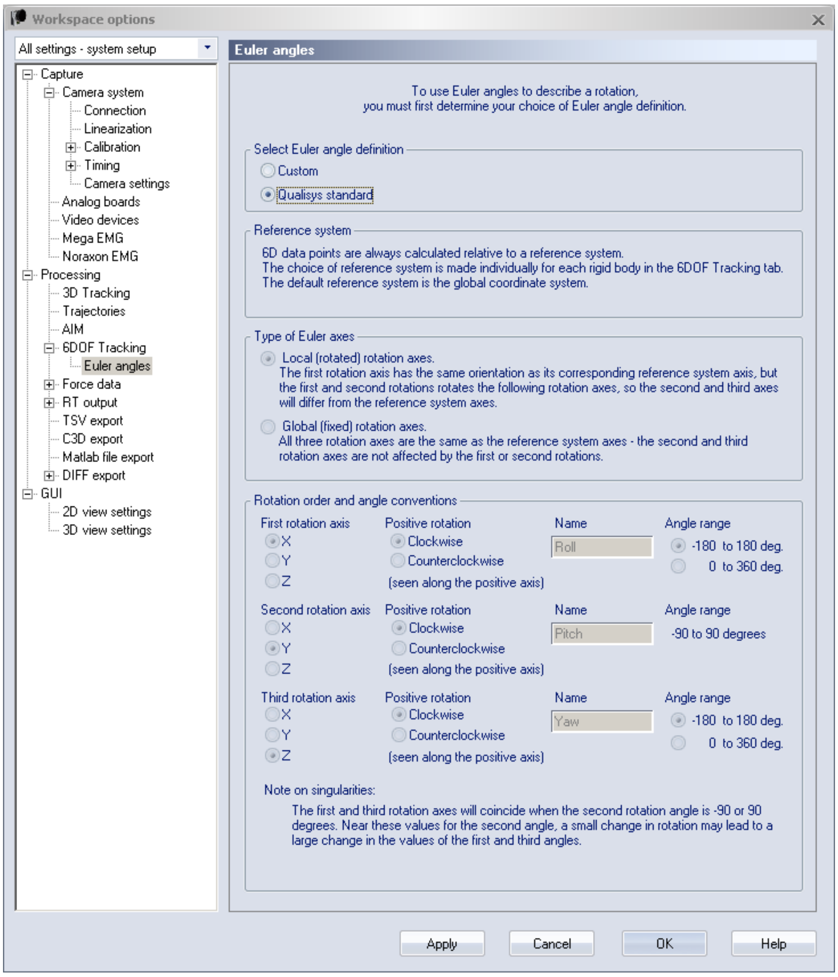
In the **Import markers** dialog labeled trajectories from the current measurement can be chosen for the points of the rigid body definition. The trajectory for each point is specified in the **Trajectory** column. Click on the arrow on the right in the column to open a list of trajectories to choose from.

If an extra point is needed, click on **Add point**. To remove a point, select it and click on **Remove point**.

Euler angles

On the **Euler angles** page you can change the definition of all transformations to and from and rotation matrix in QTM. This means that it affects among other things the transformation of the global coordinate system and the definition and measurement of 6DOF bodies.

Select Euler angle definition

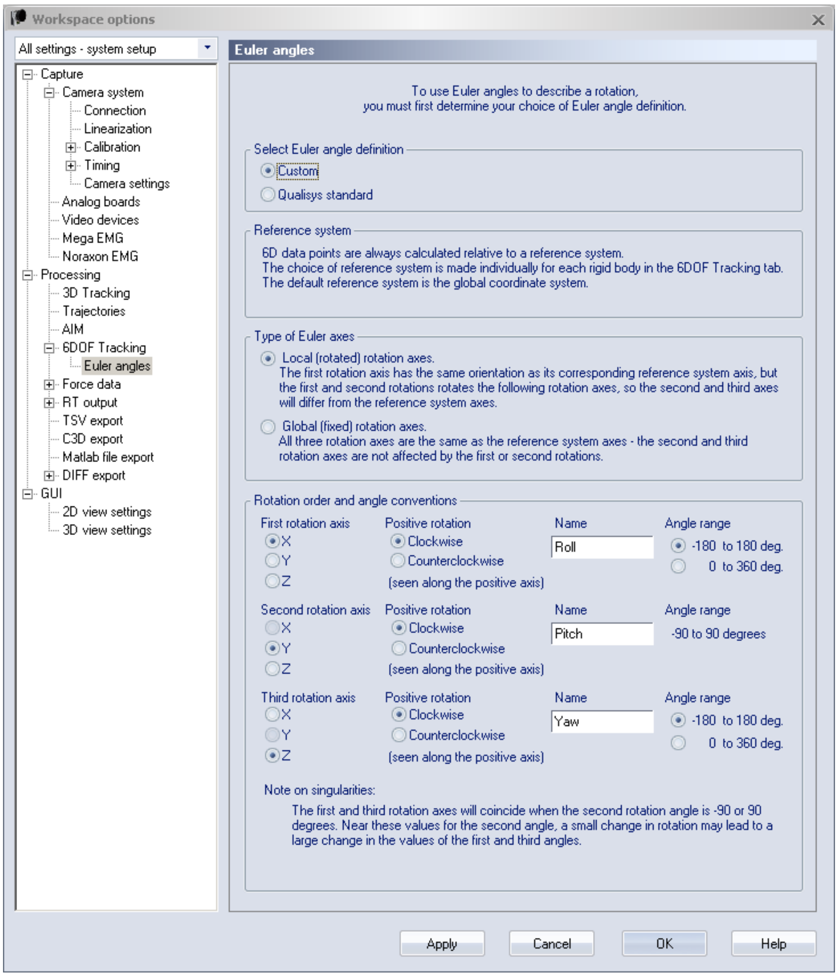


By default QTM uses the **Qualisys standard**, which is described in the chapter "Rotation angles in QTM" on page 292. The definition can also be seen when **Qualisys standard** is selected as the grayed settings under the **Definition of rotation axes** heading, see screen dump above.

Use the **Custom** setting under the **Select Euler angle definition** heading if you want another definition of the rotation angles. Change the definition under the **Definition of rotation axes** heading.

Important: If you define custom rotation angels the examples of the rotation angles in the manual does not apply. Therefore you need to have knowledge of how Euler angles work when changing this setting. However you can always go back to **Qualisys standard** angles in a file since the rotation is saved as a rotation matrix.

Definition of rotation axes



The following settings are used to define the Euler angles under the **Definition of rotation axes** heading:

Local (rotated) rotation axes
The rotations will be made round the axes of the rotated coordinate system. This means that the first rotation axis will be the same as its corresponding reference axis, defined in the **Coordinate system for rigid body data** dialog. However this will rotate the local axes and the following rotations will be made round the rotated axes which differ from the reference axes. Of course the second rotation also rotates the local coordinate system and affects the rotation axis of the third rotation.

Global (fixed) rotation axes
The rotations will be made round the axes of the coordinate system for the rigid body, which is defined in the **Coordinate system for rigid body data** dialog. The default value is the global coordinate system. When calculating the rotations this means that the first or second rotations will not affect the axes of the following rotations.

First rotation axis
Define the first rotation axis. This can be any of the three axes.

Angle range
Select the angle range of the first axis. It can be either -180° to 180° or 0° to 360°.

Second rotation axis
Define the second rotation axis. This cannot be the same as the **First rotation** axis.
Note: Because of the definition of the Euler angles it is not possible to choose range for the second axis. E.g. an angel of -10° is equal to 350°, which means that the range would be split 0° to 90° and then 270° to 350°, see more about the angles below.

Third rotation axis
Define the third rotation axis. This cannot be the same as the **Second rotation** axis. However it can be the same as the **First rotation** axis.

Angle range
Select the angle range of the first axis. It can be either -180° to 180° or 0° to 360°.

Positive rotation
For each axis you can define the direction of **Positive rotation**. It can be either **Clockwise** or **Counterclockwise** when seen along the positive direction of the axis.

Name
For each axis you can set a new name. This name will then be used everywhere in QTM.

Important: To get an unambiguous definition of the rotation there have to be two additional definitions used on the rotation angles. These two definitions are set by QTM, but the second also depends on how you define the Euler angles.

1. The rotation angles are always applied with the first rotation first and so on.
2. The rotation angles have limitations to the ranges of the angles. The first and third angle is always defined between -180° and 180° or 0° to 360°. But the middle angle depends on the Euler angle definition:

   First and third rotation round different axes

   The second rotation is always defined between -90° to 90°.
   Note: There are singularities at -90° and 90°.

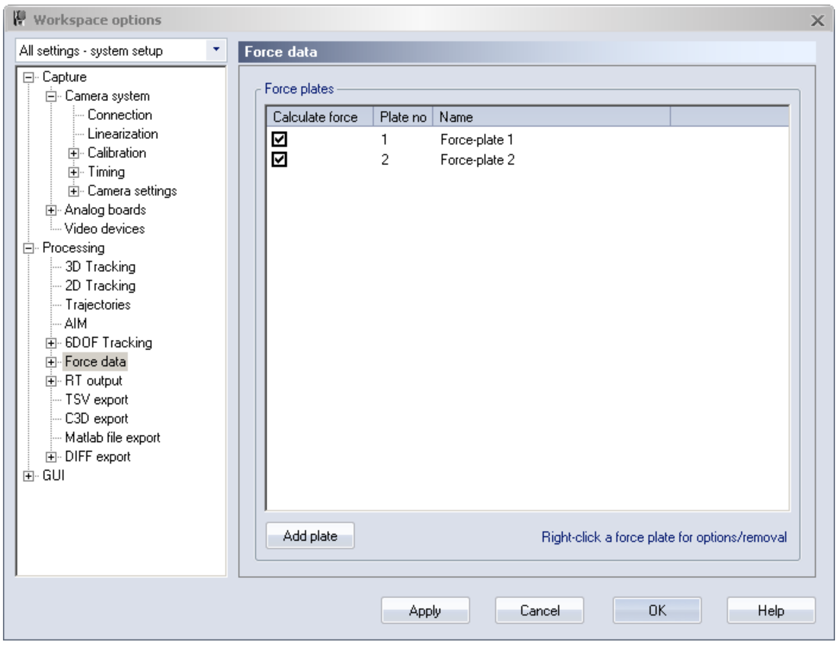
   First and third rotation round the same axis

   The second rotation is defined between 0° to 180° or 0° to -180°, depending on the order of the axes and the positive rotation. Check the text at bottom of the **Euler angle** page to find out which it is for the current definition.
   Note: There are singularities at 0° and 180° respectively at 0 to -180.

Force data

The **Force data** branch of the options tree contains settings for the force data calculation on the installed force plates. For a correct force calculation the following settings for the force plate must be specified: the dimensions, the calibration factors, the gain and the location in the coordinate system of the motion capture.

For information about how to use the force data see chapter “Force data calculation” on page 311.



Force plates

Under the **Force plates** heading on the **Force data** page there are settings for which of the force plates that the data will be calculated.

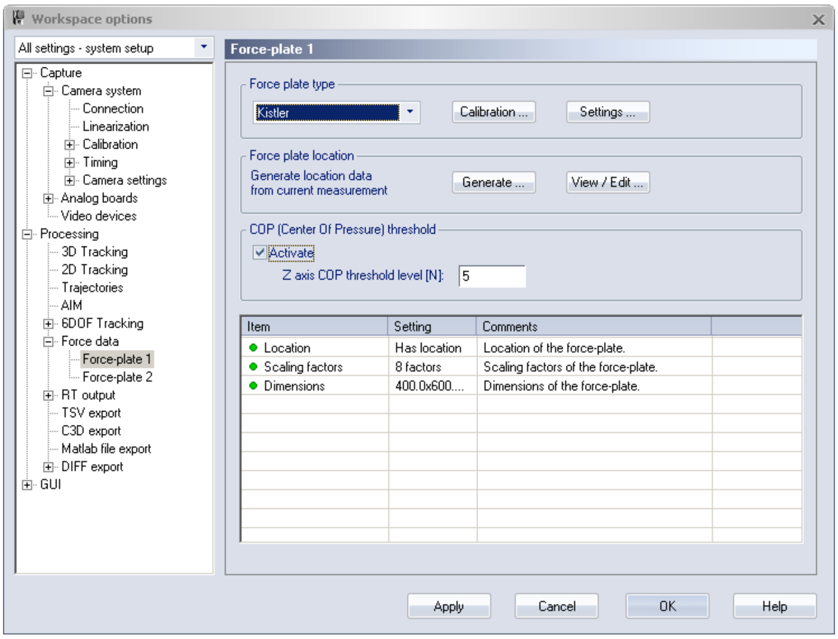
Use the **Add plate** option to add a new force plate to the list. To decide which force plates that are used in QTM, select the check box next to the force plate name in the **Calculate force** column. Right-click on the force plate to open a menu where you can **Change name** and **Remove plate**. The settings for a force plate is opened by double-clicking on its row, the settings must be entered to use the force plate see chapter “Force plate” on the facing page. For more information about force plates see chapter “How to use force plates” on page 359.

Note: The force plates that are activated will be shown in the **3D view** window even if there is no analog data.

Note: To use a force plate, the analog board that it is connected to must be selected on the **Analog boards** page.

Force plate

The settings for each force plate are found under the corresponding page. For example, settings for force plate 1 on the **Force plate 1** page.



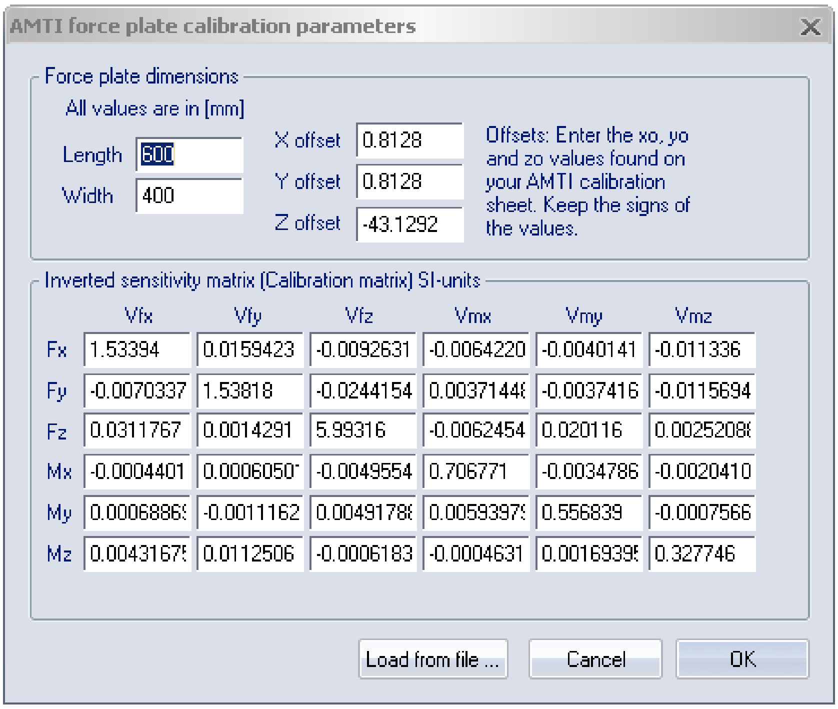
Force plate type

Under the **Force plate type** heading there are a drop-down box for the force plate type, a **Calibration** option and a **Settings** option. Select the force plate type with the drop-down box. There are five different types available: **AMTI**, **Kistler**, **Bertec**, **Portable AMTI** and **QMH**.

The **Calibration** and **Settings** options are individual for each force plate type. A description of the options can be found below for each force plate type, except for **QMH** which is a custom force plate.

AMTI force plate calibration parameters

Select the AMTI force plate type and click **Calibration** under the **Force plate type** heading to go to the **AMTI force plate calibration parameters** dialog. It contains the settings for dimensions and calibration factors of the AMTI force plate.



Force plate dimensions

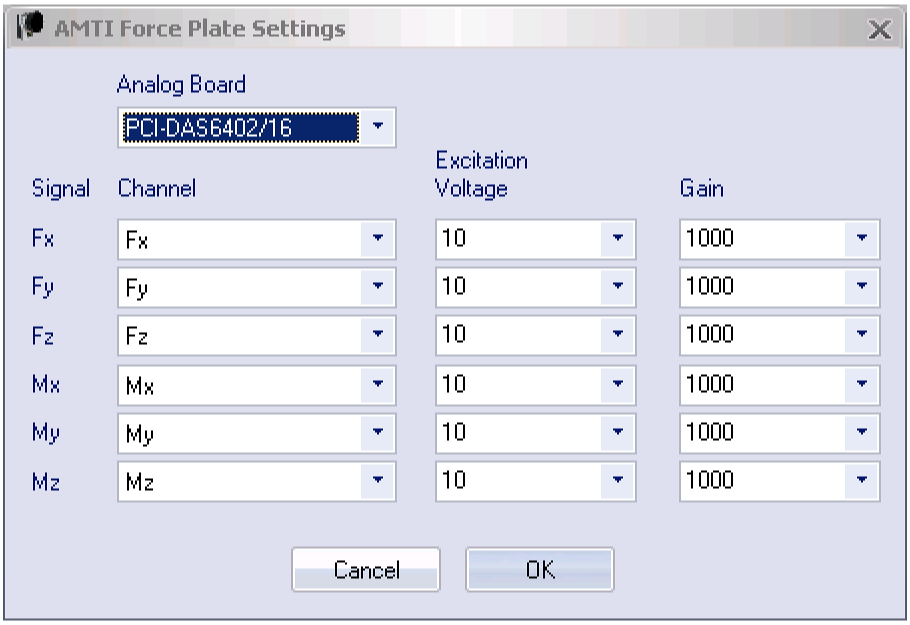
Under the **Force plate dimensions** heading you should enter the dimension parameters (in mm) for the AMTI force plate. The parameters can be found in the user manual of the AMTI force plate.

Inverted sensitivity matrix

The **Inverted sensitivity matrix** is used to calibrate the force plate. Enter the values of the inverted sensitivity matrix (in SI-units) under the **Inverted sensitivity matrix** heading. For old AMTI plates the inverted sensitivity matrix was called calibration matrix. The values can be found in the manual of the AMTI force plate or can be loaded with **Load from file** from the diskette, which is attached to the manual of the AMTI force plate.

AMTI force plate settings

Select the AMTI force plate type and click **Settings** under the **Force plate type** heading on the **Force plate** page to open the **AMTI force plate settings** dialog.



There are four settings on the dialog: **Analog board**, **Channel**, **Excitation Voltage** and **Gain**.

Select the analog board where the force plate is connected from the **Analog board** drop-down list.

With **Channel** each signal is combined with its respective analog channel.

Note: On the **Analog board (...)** page the channel names can be renamed to match the signal names.

The **Excitation Voltage** is set to the bridge excitation voltage of each channel in the AMTI amplifier.

Note: The drop-down list gives you standard values, but you can also type any value for the setting.

The **Gain** is set to the gain of each channel in the AMTI amplifier.

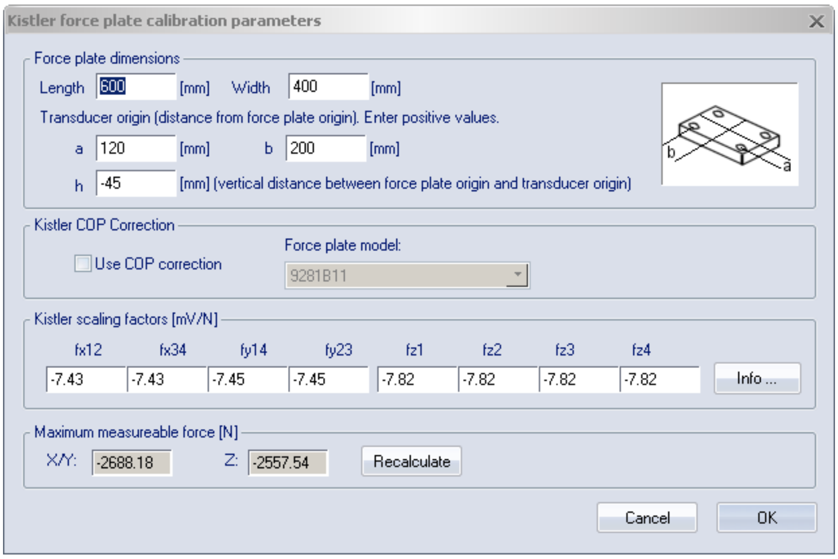
Note: The drop-down list gives you standard values, but you can also type any value for the setting.

For more information about these settings see the manual of the AMTI force plate.

Kistler force plate calibration parameters

For information on how to connect a Kistler force plate see chapter ”Connecting Kistler force plates”.

Select the Kistler force plate type and click **Calibration** under the **Force plate type** heading to go to the **Kistler force plate calibration parameters** dialog. It contains the settings for dimensions and scaling factors of the Kistler force plate.



Force plate dimensions

Under the **Force plate dimensions** heading you should enter the dimension parameters for the Kistler force plate. The parameters can be found in the user manual of the Kistler force plate.

Note: Enter the absolute values of the parameters.

Kistler COP Correction

The COP correction is a method developed by Kistler to improve the accuracy of the COP calculation. According to Kistler the error can be reduced by 3-5 times. The method is implemented in QTM from the paper "Improving COP Accuracy with Kistler Force Plates", contact Kistler for more details.

Activate the **Kistler COP correction** with the **Use COP correction** checkbox. Select the force plate model from the **Force plate model** list, the method is only available for the force plates in the list.

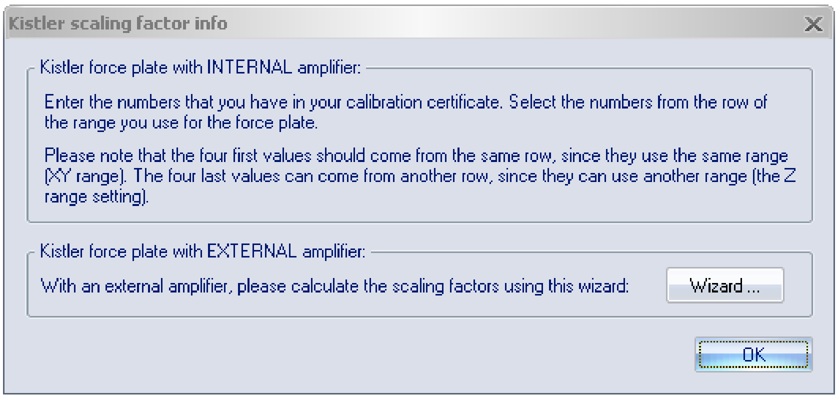
Note: The coefficients are force plate model (type number) specific. If in doubt, please contact Kistler.

Note: The coefficients apply only if the force plate is mounted on a rigid foundation according to Kistler specifications.

Kistler scaling factors

The scaling factors of the Kistler plate must be entered under the **Kistler scaling factors** heading. Enter the edit boxes with the correct values according to the descriptions below, if the wizard is used it is done automatically.

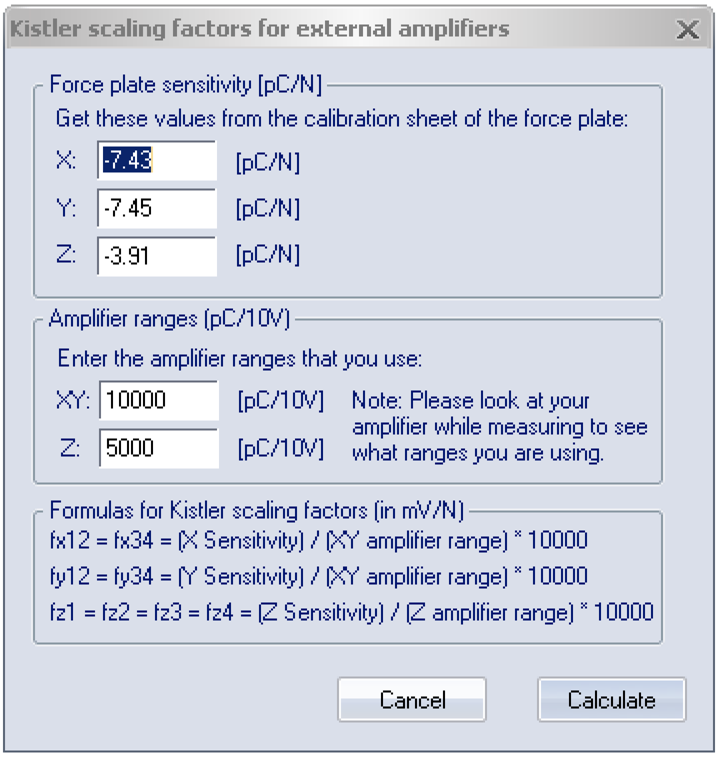
Note: The scaling factor ensures that the force data is scaled correctly. If you change the sensitivity of the force plate on the **Force plate control settings** page, these values must be updated to reflect this change. Click on **Info** to open the **Kistler scaling factor info** dialog below.



For a **Kistler force plate with internal amplifier**, enter the scaling factors in the edit boxes under the **Kistler scaling factors** heading. The scaling factors are found in the calibration certificate matrix of each force plate. Select the data from the row of the range that is used in the measurement, i.e. the ranges specified on the **Force plate control settings** page.

Note: The fx12, fx34, fy14, fy23 values is selected from the same row in the calibration matrix, since they use the same range (the XY range setting). The fz1, fz2, fz3, fz4 values on the other hand is selected from another row in the calibration matrix, since the vertical force can use another range (the Z range setting).

For a **Kistler force plate with external amplifier**, the scaling factors need to be calculated from the data found in the calibration certificate matrix of each force plate. Either use the **Wizard** or calculate the scaling factors manually, the formulas can be found in the **Wizard**. Click on wizard to open dialog below.



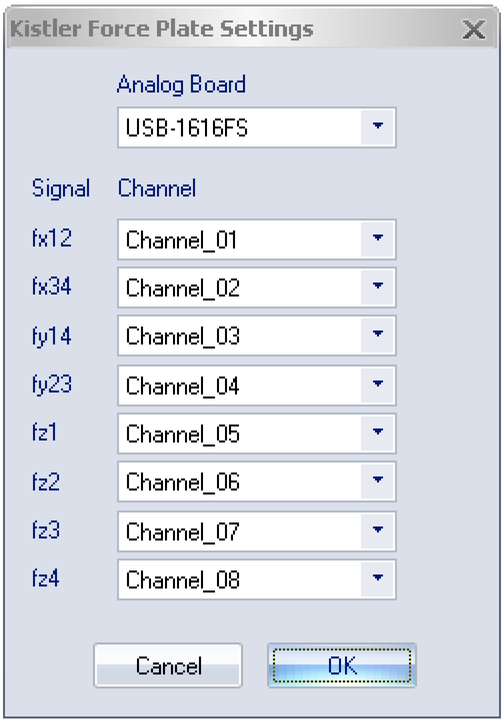
The **Force plate sensitivity** parameters are found on the calibration sheet. To get the correct value on the **Amplifier ranges** the measurement range needs to be checked, since the **Amplifier ranges** depends both on the measurement range and the ranges on the **Force plate control settings** page.

Maximum measurable force

Under the **Maximum measurable force** heading the maximum measurable forces, for the specific settings, are shown in N for the X/Y and Z directions. Click **Recalculate** to recalculate the maximum forces when a setting has been changed.

Kistler force plate settings

Select the Kistler force plate type and click **Settings** under the **Force plate type** heading on the **Force plate** page to open the **Kistler force plate settings** dialog.



First you must select the analog board where the force plate is connected from the **Analog board** drop-down list.

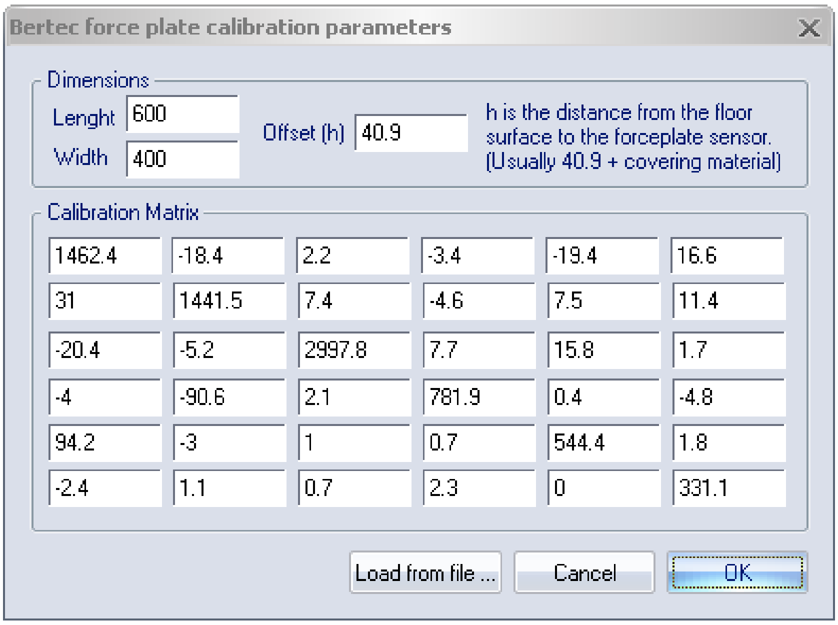
Then associate each signal from the force plate with its respective analog channel with the **Channel** settings.

Note: On the **Analog board (...)** page the channel names can be renamed to match the signal names.

For more information about these settings see the manual of the Kistler force plate.

Bertec force plate calibration parameters

Select the Bertec force plate type and click **Calibration** under the **Force plate type** heading to go to the **Bertec force plate calibration parameters** dialog. It contains the settings for dimensions and calibration factors of the Bertec force plate.



Dimensions

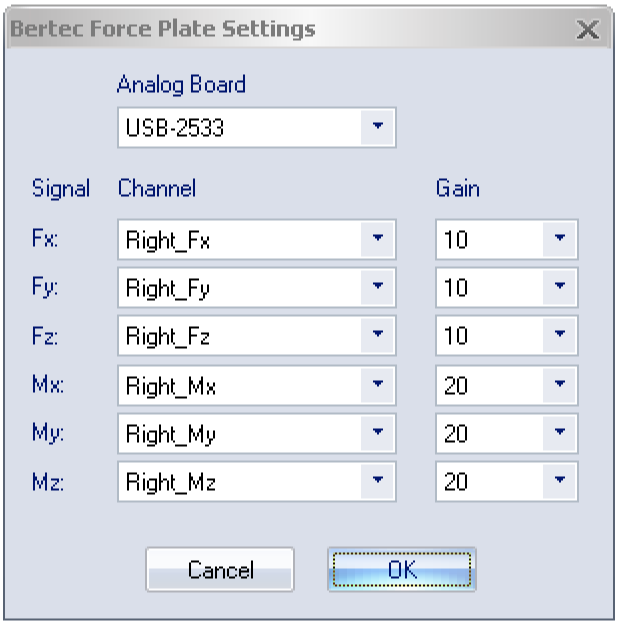
Under the **Force plate dimensions** heading you should enter the dimensions parameters for the Bertec force plate. The parameters can be found in the user manual of the Bertec force plate.

Calibration matrix

The **Calibration matrix** is used to calibrate the force plate. Enter the values of the calibration matrix, which is found in the manual of the Bertec force plate, under the **Calibration matrix** heading. Bertec often just supplies the six values of the diagonal in the matrix. The values can also be loaded with **Load from file** from the diskette, which is attached to the manual of the Bertec force plate.

Bertec force plate settings

Select the Bertec force plate type and click **Settings** under the **Force plate type** heading on the **Force plate** page to open the **Bertec force plate settings** dialog.



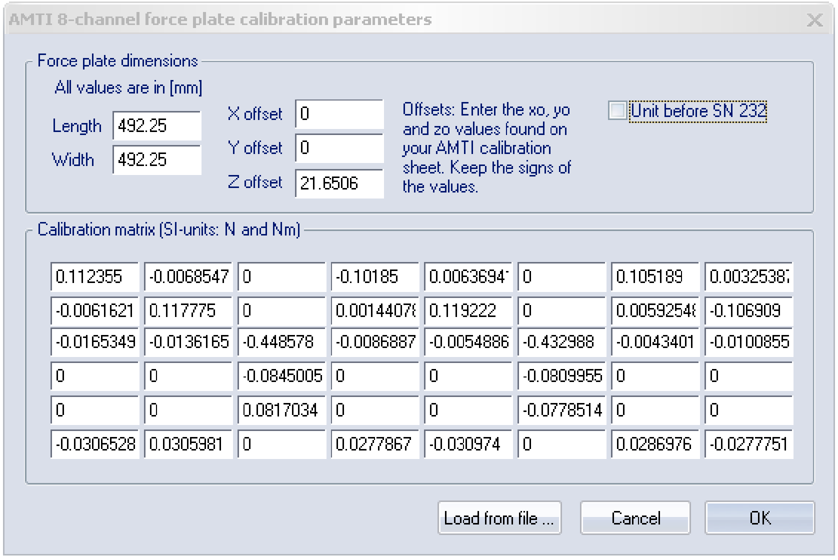
There are three settings on the dialog: **Analog board**, **Channel** and **Gain**.

> Select the analog board where the force plate is connected from the **Analog board** drop-down list.
>
> With **Channel** each signal is combined with its respective analog channel.
> Note: On the **Analog board (...)** page the channel names can be renamed to match the signal names.
>
> The **Gain** is set to the gain of each channel as selected on the Bertec amplifier.

For more information about these settings see the manual of the Bertec force plate.

AMTI 8-Channel force plate calibration parameters

Select the AMTI 8 Channel (Portable) force plate type and click **Calibration** under the **Force plate type** heading to go to the **AMTI 8-channel force plate calibration parameters** dialog. It contains the settings for dimensions and calibration factors of the AMTI portable force plate.



Force plate dimensions

Under the **Force plate dimensions** heading you should enter the dimension parameters (in mm) for the AMTI portable force plate. The parameters can be found in the user manual of the AMTI portable force plate.

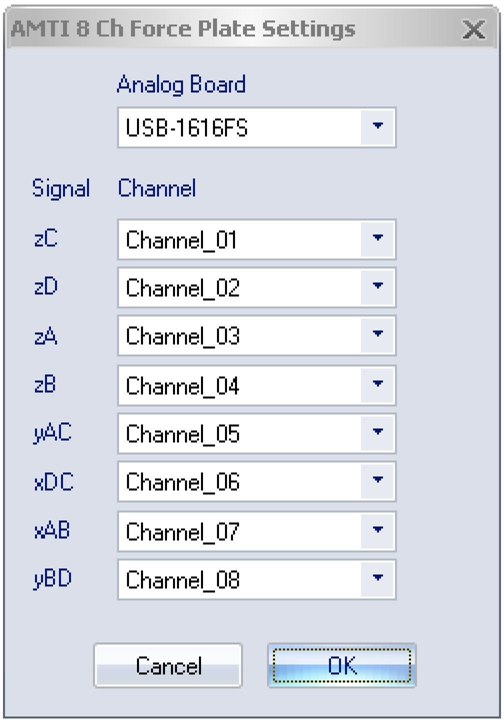
> **Old type of force plate**
> Activate the **Unit before SN 232** option, if you have a force plate with serial number lower than 232. These force plates have a "zero" level of 2.5 V. When calculating the force in a file, the data is always zeroed with the first 10 frames of the measurement. This is because the error is quite large and because of the 2.5 V offset you cannot use the standard option to remove the offset.

Calibration matrix

The **Calibration matrix** is used to calibrate the force plate. Enter the values of the calibration matrix (in SI-units) under the **Calibration matrix** heading. The values can be found in the manual of the AMTI force plate or can be loaded with **Load from file** from the diskette, which is attached to the manual of the AMTI portable force plate.

AMTI 8 Ch force plate settings

Select the AMTI 8 Channel (Portable) force plate type and click **Settings** under the **Force plate type** heading on the **Force plate** page to open the **AMTI 8 Ch force plate settings** dialog.



First you must select the analog board where the force plate is connected from the **Analog board** drop-down list.

Then associate each signal from the force plate with its respective analog channel with the **Channel** settings.

Note: The channel names can be renamed to match the signal names on the **Analog board (...)** page.

For more information about the signals see the manual of the AMTI portable force plate.

Force plate location

To be able to view the force vectors in the same coordinate system as the motion capture, the location of the force plate must be specified. The settings are the same for all force plate types and are done under the **Force plate location** heading on the **Force plate** page. The force plate location will be visualized as a purple square in the **3D view** window.

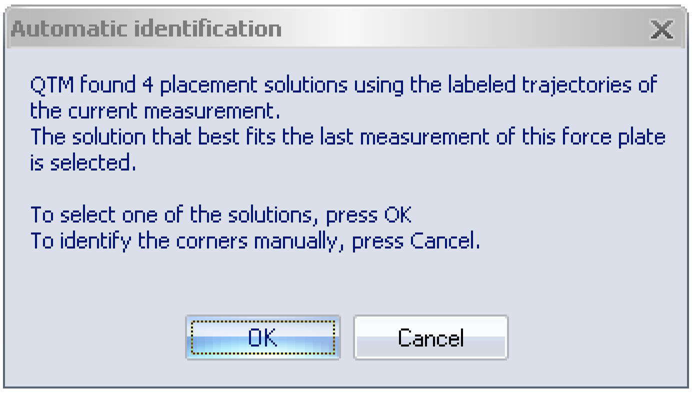
There are two ways to do this: by a motion capture or manually.

1. To specify the location by a motion capture, place a marker on top of each of the four corners of the force plate.

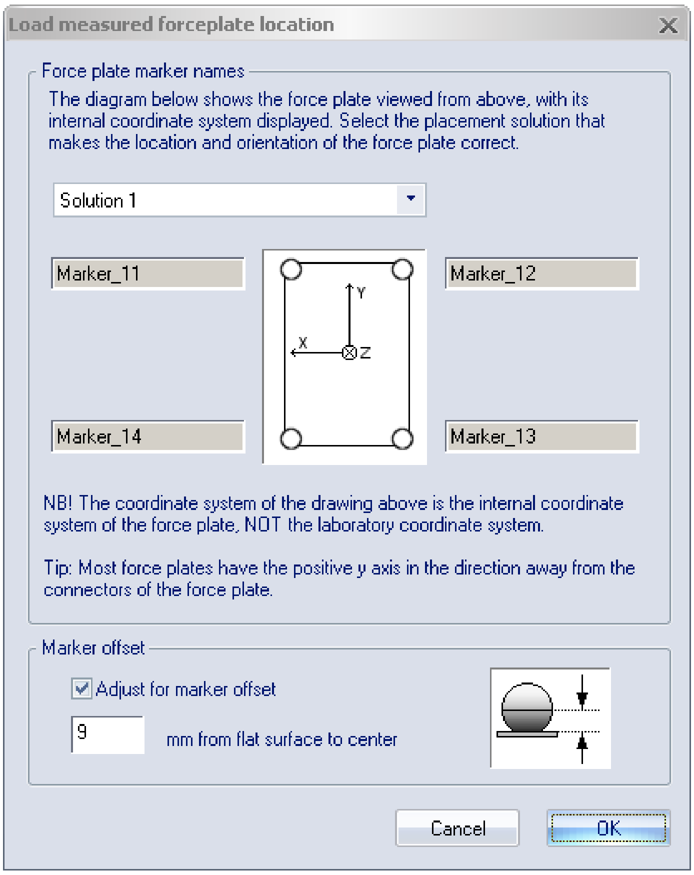
   Make a 1 second motion capture of these markers.

   Identify the markers, you can give them any label, and keep the capture file open.

   Open the **Force plate** page and click **Generate**. QTM tries to identify the corners of the force plate by comparing them to the width and length of the plate (as entered in the **Force plate calibration parameters** dialog). A dialog box like the one below is displayed:



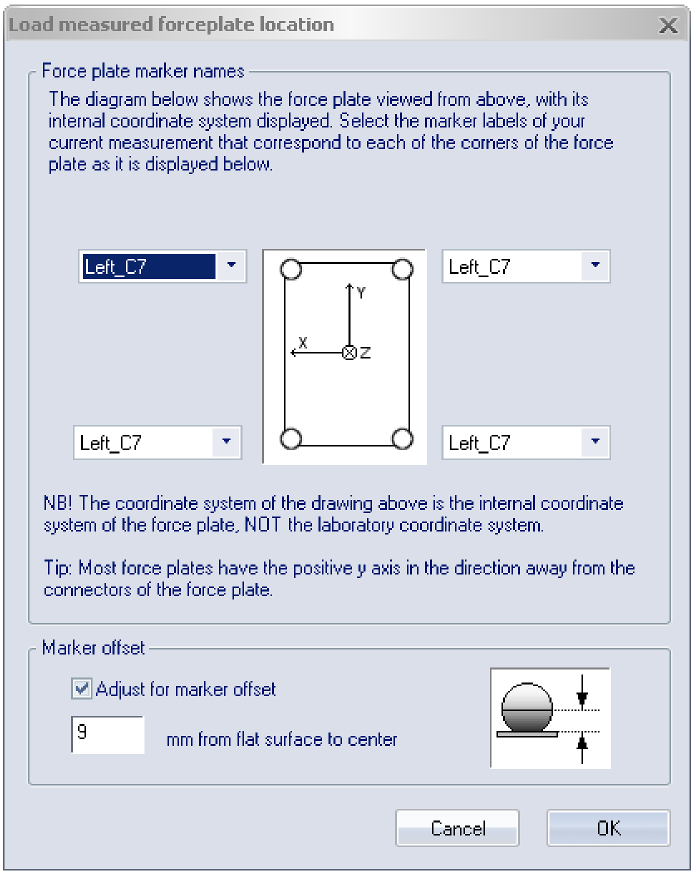
Click **OK** to open **Load measured force plate location** dialog, see below, and select one of the solutions found by QTM. Click **Cancel** to select the markers' locations manually, see further down. Try to make sure that the orientation is correct. It is recommended to make a test measurement after the location process to see that the orientation is correct. If the force arrow is pointing downwards, you can use the **Rotate 180 degrees** option in the **Force plate location** dialog, shown below.



When spherical markers are used, the level of the force plate in the vertical direction will be the centers of the markers. If the level of the force plate should be exactly in level with the actual force plate, use the **Adjust for marker offset** setting in the **Load measured forceplate location** dialog. You can also click **View/Edit** in the **Force plate settings** dialog and adjust the vertical position. E.g. for the following standard markers the vertical level should be lowered according to following table:

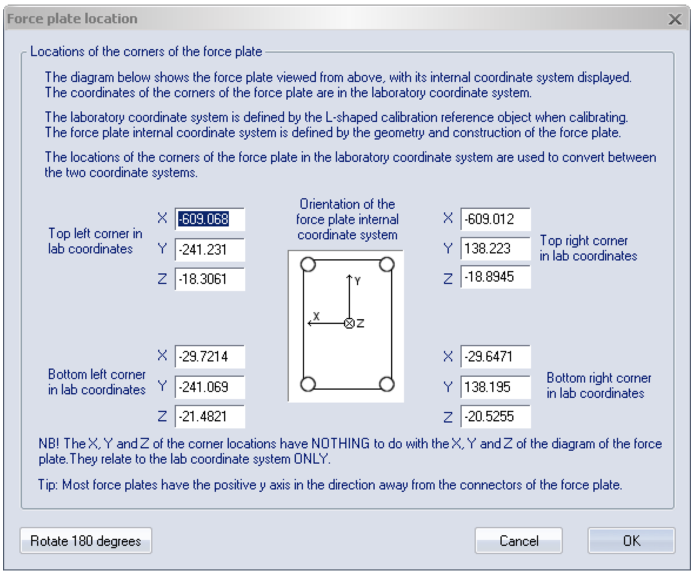
Marker size	Adjustment
12 mm	5 mm
19 mm	8 mm
30 mm	13 mm

Note: If QTM cannot find a solution among the labeled trajectories or if you click **Cancel** in the **Automatic identification** dialog, almost the same dialog will appear but you must select each corner manually, see below. in this case it is especially important to test that the force plate location is correct before making the measurements.



2. To manually specify the location click **View/Edit**. Enter the X, Y and Z coordinate (in mm) of the four corners of the force plate. The coordinates should be in the coordinate system of the motion capture (lab coordinates).

   Make sure that the orientation is correct. Use the internal coordinate system of the force plate, shown in the dialog, to determine the orientation of the force plate corners. Most force plates have the positive y axis in the direction away from the connectors of the force plate.



COP (Center Of Pressure) threshold

The Center Of Pressure (COP) is the position of the center of the pressure on the force plate. It is calculated by QTM from the data of the pressure sensors in the corners of the force plate.

The **COP (Center Of Pressure) threshold** heading on the **Force plate** page contains settings for the COP threshold. It is a filter, which disables the calculation of COP as long as the force is less than the threshold.

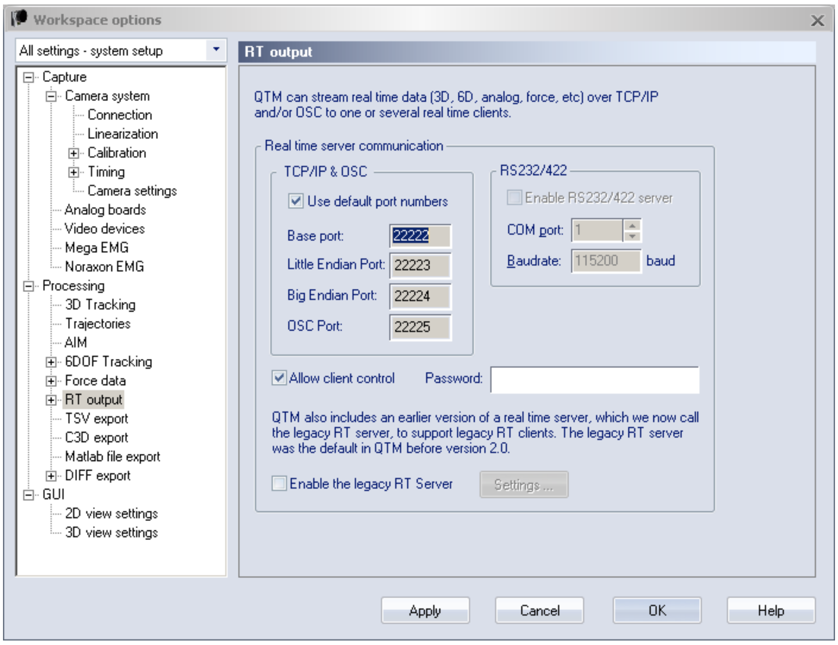
Select the **Activate** check box to activate the COP threshold filter. Enter the **Z axis COP threshold level** in Newton to disable the calculation of the COP. The Z axis is in the force plate coordinate system, which means that it is always the vertical force which is used in the filter. This is because the horizontal forces can be very small but still correct.

Force plate settings status window

At the bottom of the **Force plate** page there is a **Settings status** window, which shows the status of the settings of the force plate. It uses the same notification icons as under the **Camera system settings** heading on the **Camera system** page. The three settings that are shown are **Location**, **Scaling factors** (**Calibration** for the AMTI force plates) and **Dimensions**.

RT output

With the real-time output function any tracked data (3D or 6DOF) and also analog and force data can be sent to another computer via TCP/IP. The RT server is always running so that a program can acquire the RT data. The settings for the real time server is found under the **Real time server communication** heading.



Real time server communication

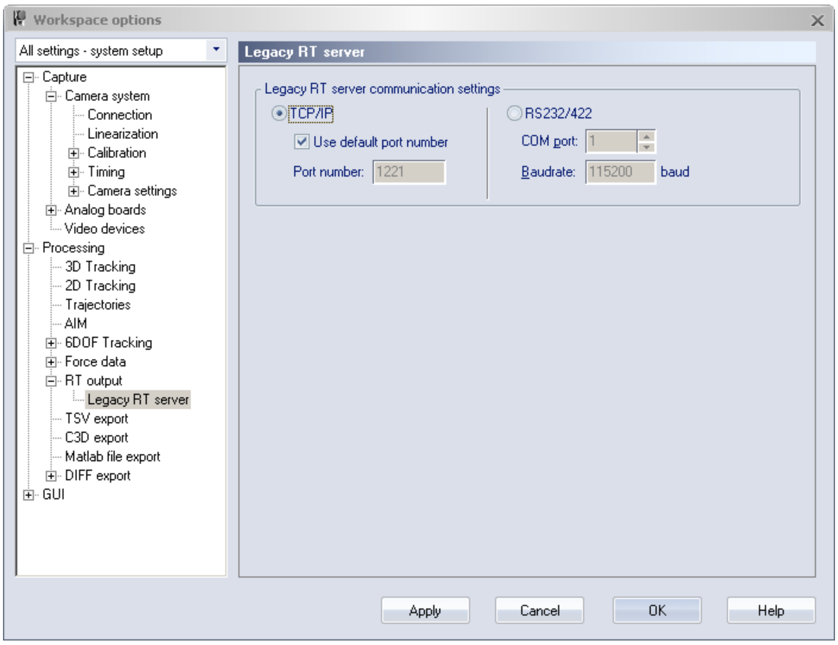
The setting that can be changed for the RT output is the TCP/IP and OSC port numbers. Uncheck the option **Use default port numbers** to set other port numbers. The port number that can be changed is the **Base port** number, which by default is 22222. The other ports used in the RT protocol are then set from the **Base port** number, for a description of the different ports see the SDK for the RT output.

Then you can also activate the client control with the **Allow client control** option. Activate the option to enable RT clients to control QTM (master mode), for example with the iOS apps. If you set a **Password**, that password must be sent from the RT client for it to run in master mode. The RT clients can always connect to QTM and receive the RT output (slave mode), which is the only mode that is available if **Allow client control** is deactivated.

Note: Only one RT client at a time can be in master mode to control QTM.

Enable the old RT server that could only output 6DOF data with **Enable the legacy RT server**. Click on **Settings** to go to the **Legacy RT server** page, see chapter “Legacy RT server” on next page.

Legacy RT server

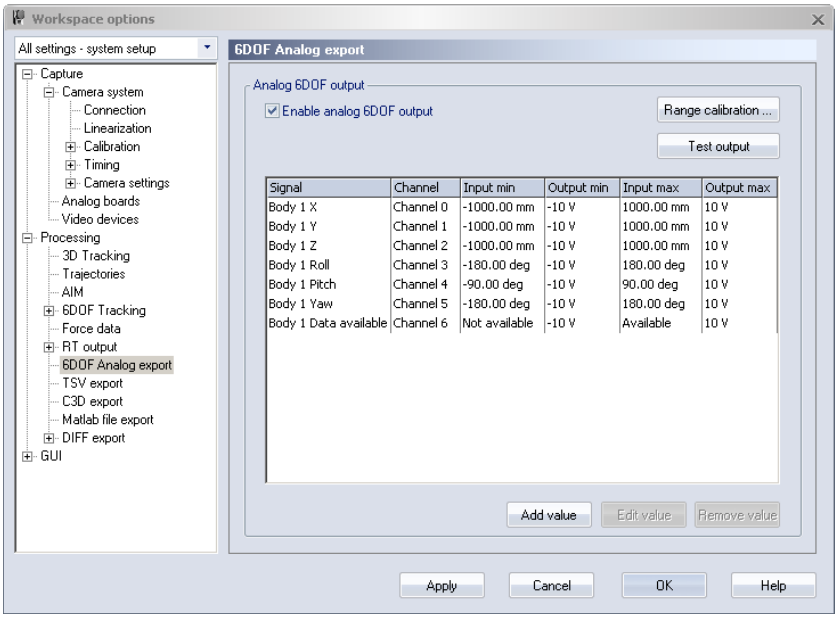


Under the **Legacy RT server communication settings** heading there are settings for the communication of the old RT server that can only handle 6DOF data. For this RT server the data is accessed in the remote computer with a DLL file.

The communication can be done either via **TCP/IP** or **RS232/422**, where the **TCP/IP** is the much faster option. For **TCP/IP** the port number is always set to 1221. While the number (**COM port**) and **Baudrate** can be changed for the **RS232/422** setting.

6DOF analog export

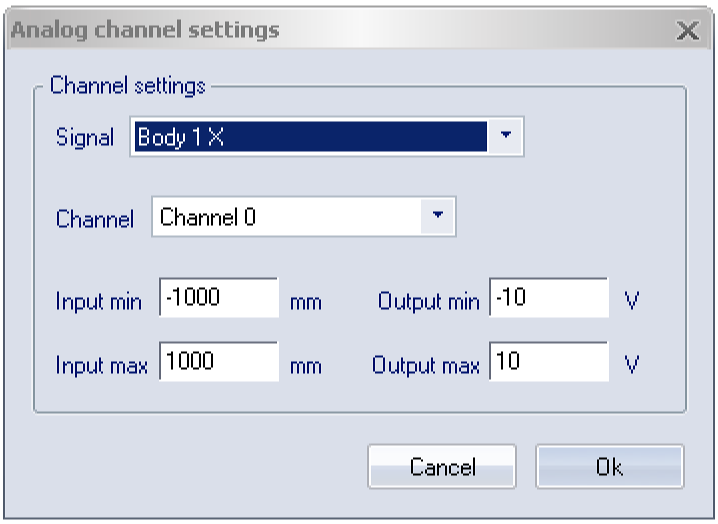
The **6DOF analog export** page is only available if an analog output board (PCI-DAC6703) is installed in the measurement computer. With the analog export the information about 6DOF bodies' positions can be used in feedback to an analog control system. Select the **Enable analog 6DOF output** option to enable the output of the board. The output will continue as long as the option is selected and the 6DOF body is being tracked.



The list on the **6DOF Analog export** page contains the wanted signals and their respective settings. Use **Add value** or double-click in the empty area to add a new signal, see chapter "Analog channel settings" below. Use **Edit value** or double-click on the signal to change the settings of the selected signal. With **Remove value** the selected signals are removed from the list.

Before using the analog export you should set the **Range calibration**. Use the **Test output** option to test the output of the analog board.

Analog channel settings



When clicking on **Add value** or **Edit value** the **Analog channels settings** dialog is displayed. In the dialog the following settings can be set:

Signal
The data in QTM that is used for the output signal. For each 6DOF body on the **6DOF bodies** page there are seven available outputs: **X**, **Y**, **Z**, **Roll**, **Pitch**, **Yaw** and **Data available**. **Data available** shows whether the 6DOF body is visible or not.
Note: The rotation angles will change if you change the Euler angles definitions.

Channel
The channel on the analog output board that will be used for the signal. Each channel can only have one signal assigned to it.

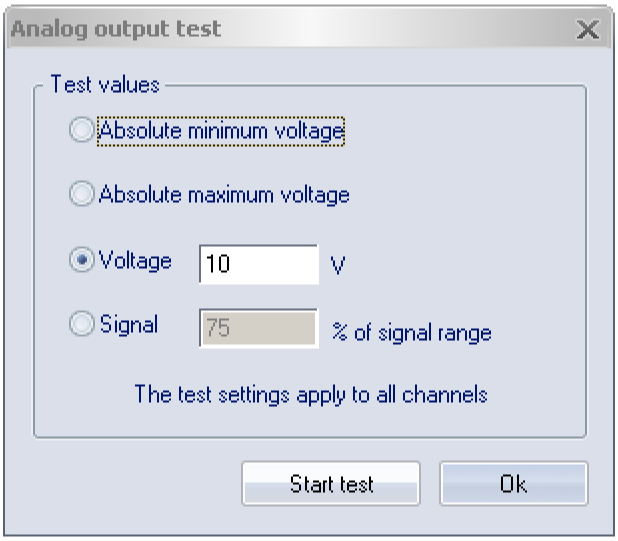
Input min/Input max
The minimum respectively the maximum value (mm or degrees) of the input data. If the input data is equal or smaller than the **Input min** the output of the channel is equal to the **Output min**. If the input data is equal or larger than the **Input max** the output of the channel is equal to the **Output max**.
Note: For the three rotation angles the maximum input ranges depend on the ranges of the respective angle.

Output min/Output max
The minimum respectively the maximum value (V) of the output on the channel.

Note: **Data available** has two positions **Available** and **Not available** instead of the input and output settings. Set the value in V which will be on the channel depending on whether the 6DOF body as seen or not.

Test output

When clicking on **Test output** the dialog below is opened.



In the dialog four tests can be performed to test the output of the channels:

Absolute minimum voltage
The output of all channels are set to the minimum value of the analog board. This value should be entered on the **Analog output range calibration** dialog.

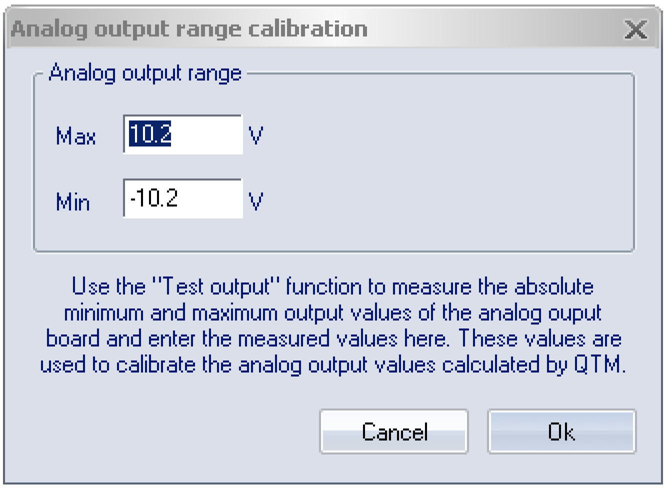
Absolute maximum voltage
The output of all channels are set to the maximum value of the analog board. This value should be entered on the **Analog output range calibration** dialog.

Voltage
The output of all channels are set to the specified voltage.

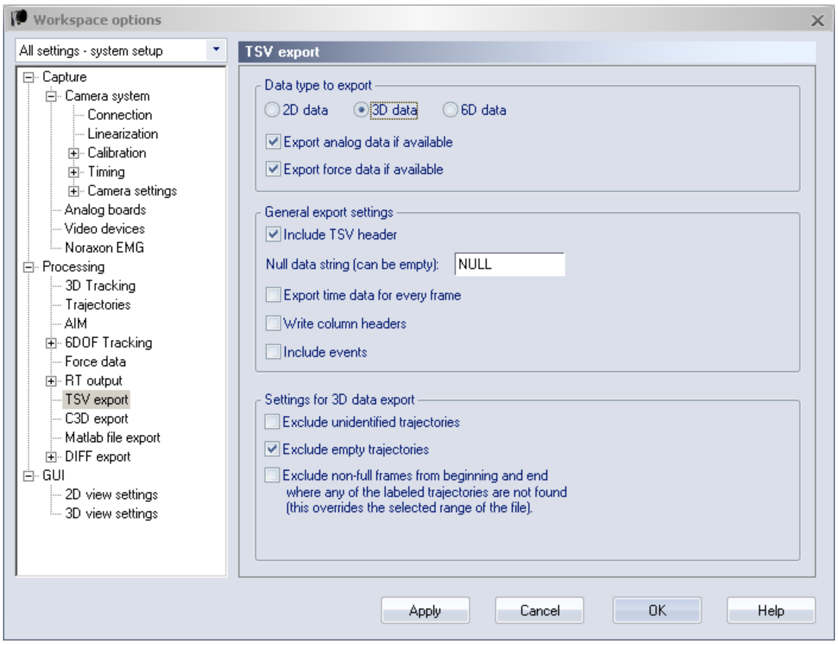
Signal % of signal range
The output of all channels is set to the specified percentage of the channel's specified output range. If the channel is not used the output will be 0 V.

Range calibration

In the **Analog output range calibration** dialog the range of the specific board is entered to calibrate the output of the channels. The maximum and minimum values can be measured with the **Test output** option.



TSV export



On the **TSV export** page there are settings for the TSV export, for information about the TSV export see chapter "Export to TSV format" on page 314.

Data type to export

Under the **Data type to export** headings there are settings for which data types to export. The settings are:

2D data tsv export
Export the 2D data of the capture file.

3D data tsv export
Export the 3D data of the capture file.

6D data tsv export
Export the 6D data of the capture file.

Then you can also select whether to include analog and force data in the export with the options, **Export analog data if available** and **Export force data if available**.

General export settings

The settings under the **General export settings** heading are applied to all TSV exports. The settings are:

Include TSV header
Include the TSV header with information about the capture file in the exported file.

Null data string
Set the data string for empty data frames in a trajectory, e.g. '0', '-' or 'NULL'. Do not write the quotation marks. The null data string can be empty, i.e. it does not need to contain a character.

Export time data for every frame
Include two columns with frame number and measurement time for each frame in

the exported TSV file. The time data will be entered in the first two columns of the file. If the SMPTE timecode exist, it will be added as a third column after the measurement time.

Note: For Non-periodic external timebase you will not be guaranteed to get the correct timestamps, because the actual frequency of the measurement is unknown.

Write column headers
Include a header over each column of data, which describes the data in that column. E.g. for 3D data the three different positions (Toe X, Toe Y and Toe Z)

Include events
Include the events in the 2D, 3D or 6DOF TSV file.

Settings for 3D data export

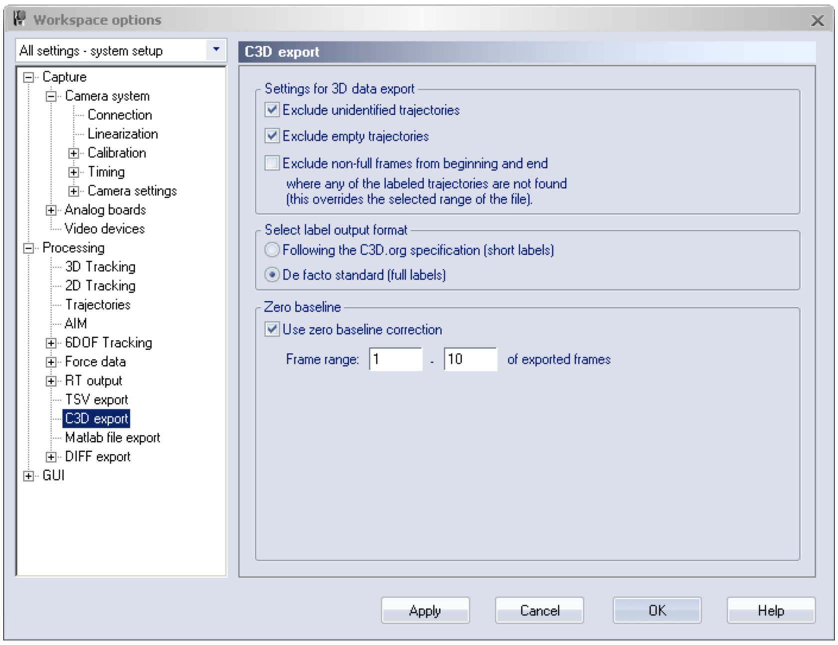
The settings under the **Settings for 3D data export** heading are only applied to 3D data. The settings are:

Exclude unidentified trajectories
Exclude unidentified trajectories from the exported file.

Exclude empty trajectories
Exclude empty trajectories without data (e.g. a new label) from the exported file.

Exclude non-full frames from beginning and end where any of the labeled trajectories are not found
Exclude completely empty frames of the labeled trajectories, in the beginning and the end of the measurement.

C3D export



On the **C3D export** page there are settings for the C3D export, for information about C3D export see chapter “Export to C3D format” on page 320.

Settings for 3D data export

The settings under the **Settings for 3D data export** heading are only applied to 3D data. The settings are:

Exclude unidentified trajectories
Exclude unidentified trajectories from the exported file. If unidentified trajectories are included they will not have a name in the C3D file.

Exclude empty trajectories
Exclude empty trajectories without data (e.g. a new label) from the exported file.

Exclude non-full frames from beginning and end where any of the labeled trajectories are not found
Exclude completely empty frames of the labeled trajectories, in the beginning and the end of the measurement.

Select label output format

Under the **Select label output format** heading you can change the format of the C3D file with the following two settings:

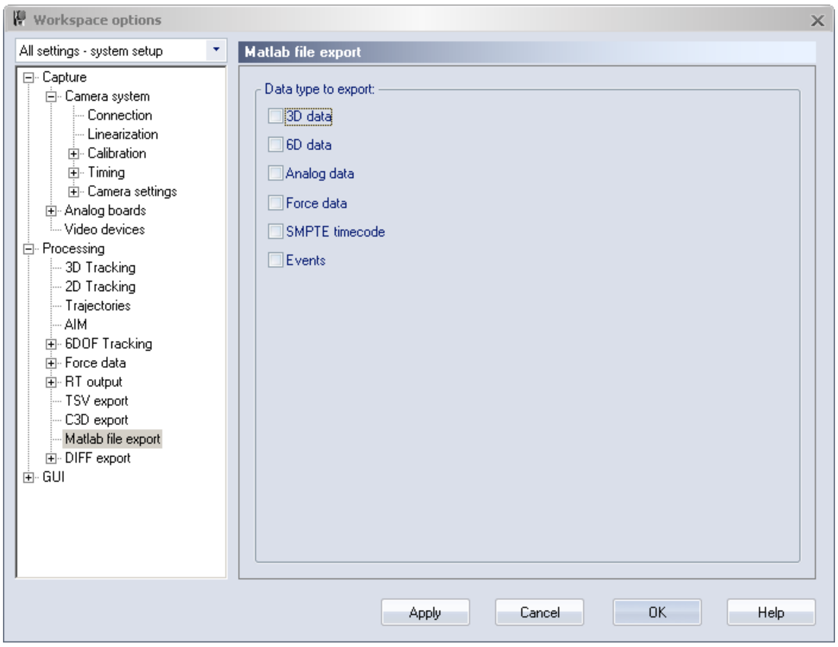
Following the C3D.org specification (short label)
Use the C3D.org specification, which uses short labels.

De facto standard (full labels)
Use full labels, i.e. the full names for the trajectories and analog channels will be included in the file.

Zero baseline

The **Zero baseline correction** is a setting used on force data in the C3D file. If activated the program that opens the C3D file, e.g. Visual3D, will zero the force data using the specified **Frame range**. This means that you cannot have any force on the force plate during these frames. The frames are set in relation to exported frames and therefore if the measurement range is changed the frames used to zero the baseline is also changed.

Matlab file export



The **Matlab file export** page contains settings for the export to MAT files. For information about MAT file export see chapter "Export to MAT format" on page 322. The export directly to Matlab is not changed by these settings. Select which data to export with the **Data type to export** options. The following data can be selected.

3D data
All of the 3D data will be included, both labeled and unidentified.

6D data

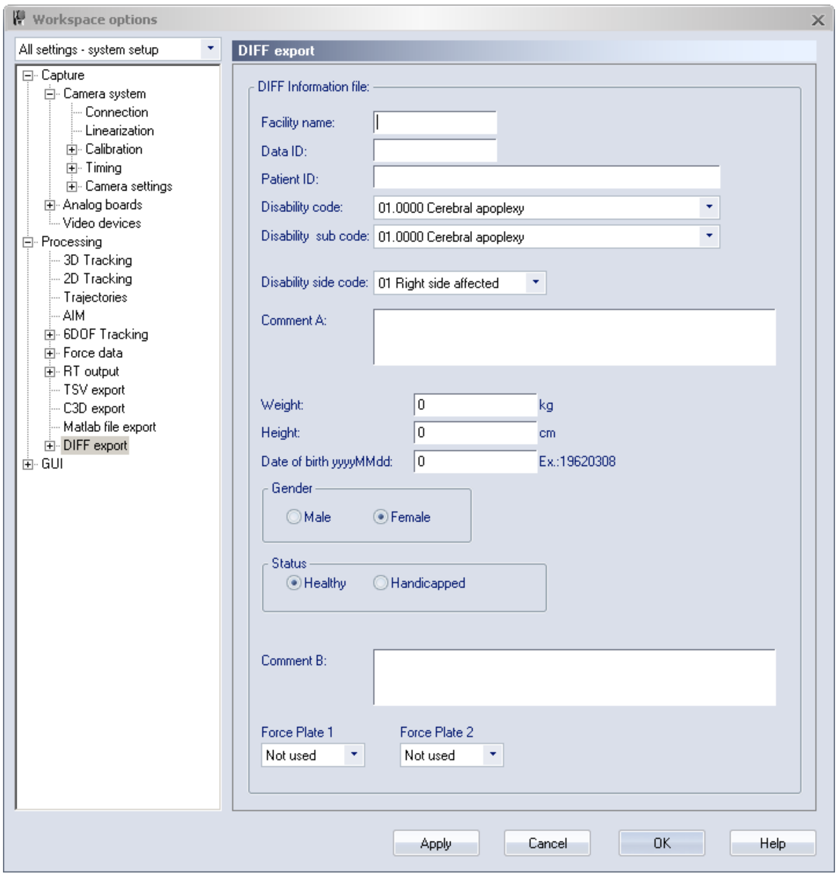
Analog data
All of the analog data will be included, both data from analog boards and EMG.

Force data

SMPTE timecode

Events

DIFF export



On the **DIFF export** page there are settings for the subject information that are exported in the DIFF file. For the label settings see chapter “G1R” on next page, “G1L” on page 215, “G2R” on page 216 and “G2L” on page 217. The DIFF format is mainly used in Japan for transferring data for Gait Analysis, for the definition refer to the DIFF Data Interface File Format (DIFF) User's Manual.

For information on how to export a DIFF file in QTM see chapter “Export to DIFF format” on page 321.

The following subject information can be entered for the DIFF file.

Facility name
The name of the facility, max number of characters 8.

Data ID
The data ID is the filename excluding the two last characters before the period, max number of characters 8.

Patient ID
A ID for the patient, max number of characters 50 (equivalent to 25 kanji characters).

Disability code
The thirteen main disability codes. The disability code represents the classification of the dysfunction according to the Functional Independence Measure (FIM). For subjects with multiple disability, the disability code which is the main cause of walking dysfunction should be entered.

Disability sub code
Sub codes to the selected **Disability code**.

Disability side code
Code for which side is affected by the disability.

Comment A
An overall comment to multiple trials of walking. The first 9 characters of **Comment A** is the **Disability code** and **Disability side code**. Max number of characters 245 (equivalent to 122 kanji characters).

Weight
Weight of the subject in kg.

Height
Height of the subject in cm.

Date of birth
Date of birth of the subject, in the format yyyyMMdd, e.g. 19630326.

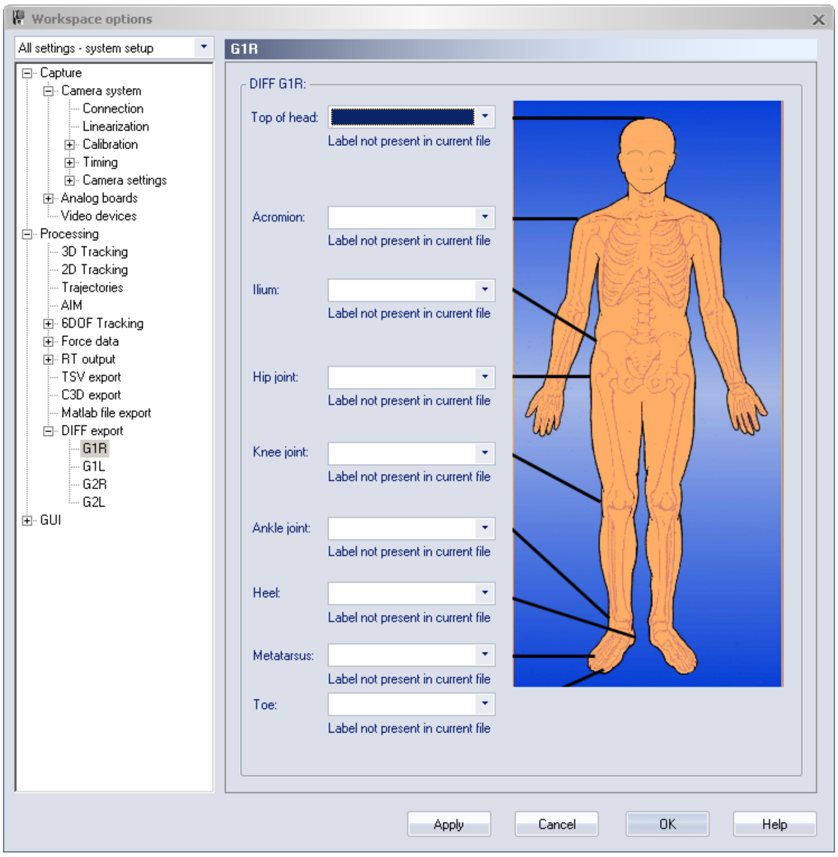
Gender
Gender of the subject.

Status
Status of the subject, **Healthy** or **Handicapped**.

Comment B
A comment related to the current trial of walking, for example rapid walking. Max number of characters 254 (equivalent to 127 kanji characters).

Force Plate 1...
Select which foot hits the force plate.

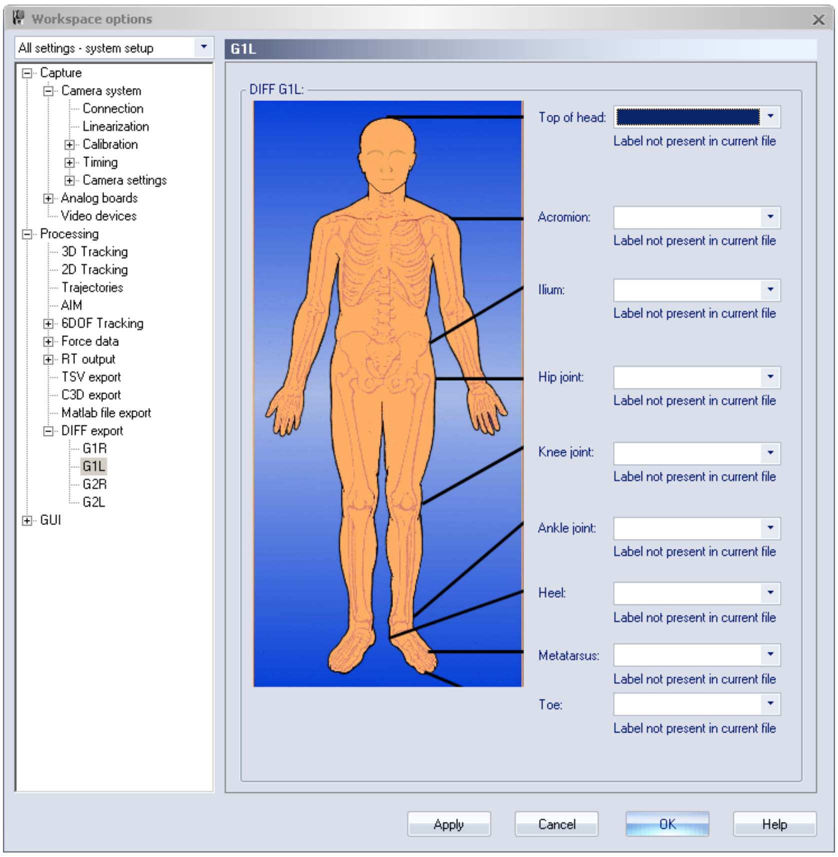
G1R



The **G1R** page contains the settings for the labels on the right side of the full body. You can specify an optional number of the labels, any labels that are not present in the file will be empty in the DIFF export. The maximum length of the label name is 128 characters.

To select the names of the labels you must have the labels in the **Labeled trajectories** list. Either open a saved file or apply an AIM model in RT to get the labels in the drop-down lists.

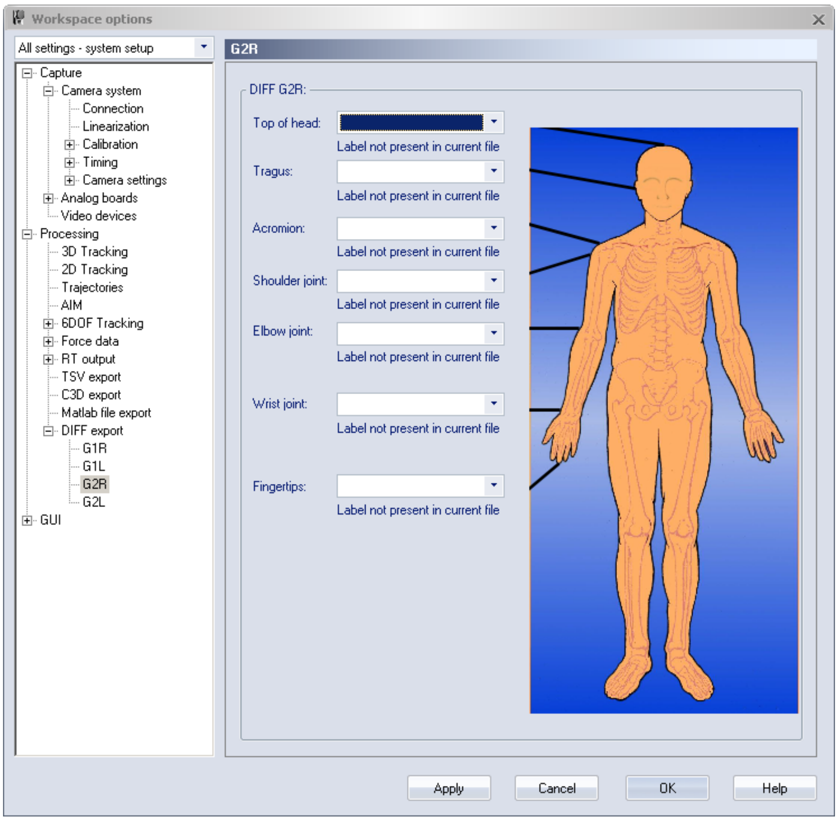
G1L



The **G1L** page contains the settings for the labels on the left side of the full body. You can specify an optional number of the labels, any labels that are not present in the file will be empty in the DIFF export. The maximum length of the label name is 128 characters.

To select the names of the labels you must have the labels in the **Labeled trajectories** list. Either open a saved file or apply an AIM model in RT to get the labels in the drop-down lists.

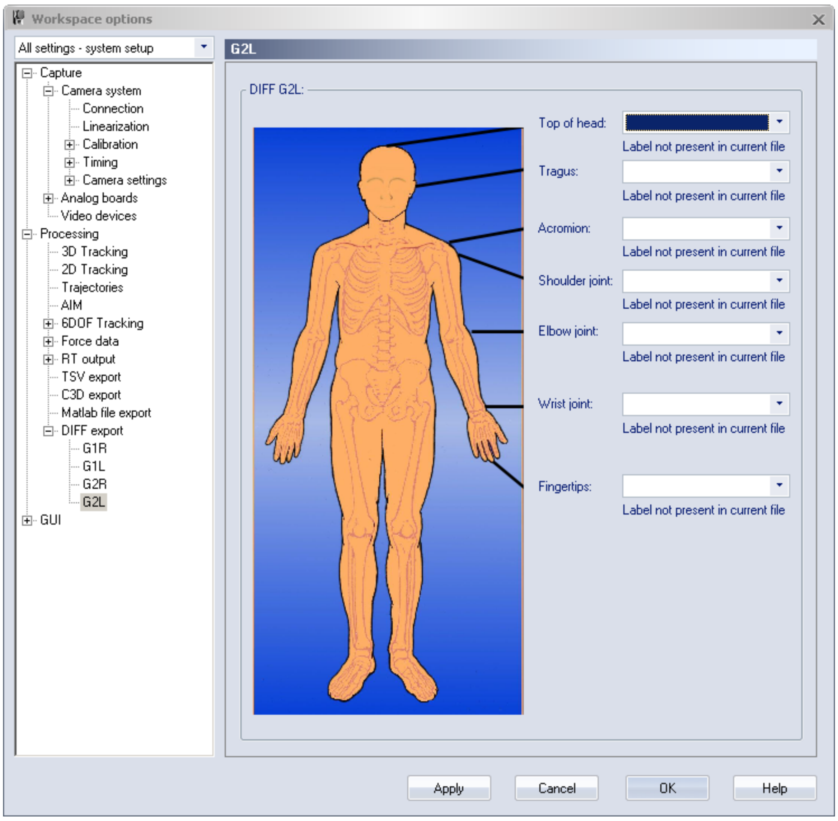
G2R



The **G2R** page contains the settings for the labels on the right arm. You can specify an optional number of the labels, any labels that are not present in the file will be empty in the DIFF export. The maximum length of the label name is 128 characters.

To select the names of the labels you must have the labels in the **Labeled trajectories** list. Either open a saved file or apply an AIM model in RT to get the labels in the drop-down lists.

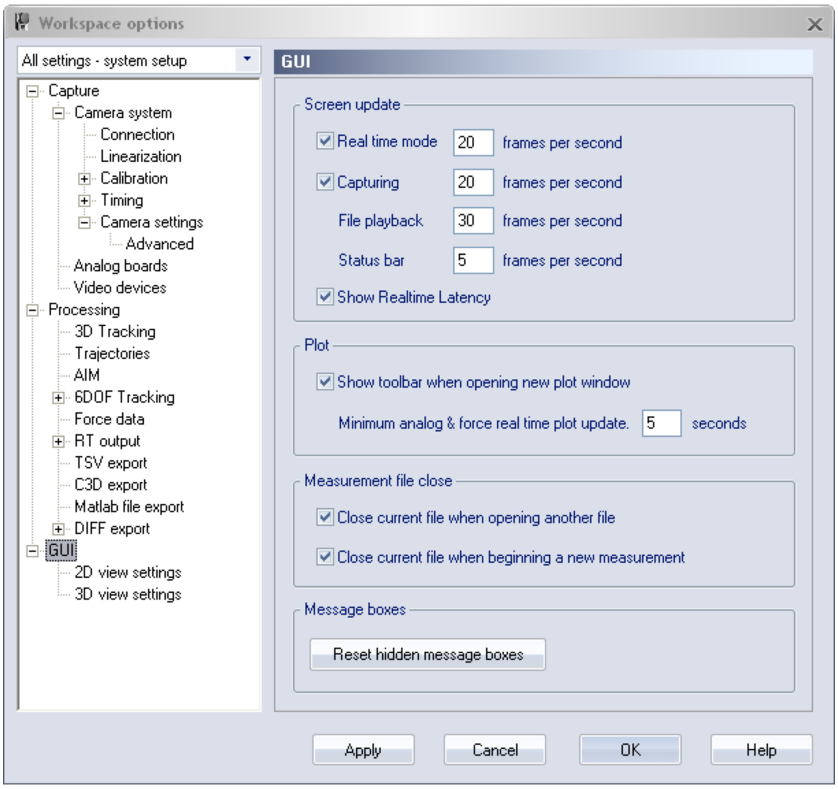
G2L



The **G2L** page contains the settings for the labels on the left arm. You can specify an optional number of the labels, any labels that are not present in the file will be empty in the DIFF export. The maximum length of the label name is 128 characters.

To select the names of the labels you must have the labels in the **Labeled trajectories** list. Either open a saved file or apply an AIM model in RT to get the labels in the drop-down lists.

GUI



The **GUI** branch contains display related settings. All of the GUI settings are applied both in RT and on files. On the **GUI** page there are settings for the **Screen update** and for **Close previous measurement file**. Except for the GUI page there are **2D view settings** and **3D view settings** in the branch.

The **Screen update** options can be used to optimize the performance of the computer. Most of the time 15 Hz is enough for the screen update as the eye cannot register movements faster than 25 Hz.

Real time mode
Set the frequency for the screen update during real time (preview).

Capturing
Set the frequency for the screen update during capturing.

File playback
Set the frequency for the screen update during file playback.

Status bar
Set the frequency for the update of the numbers in the status bar.

Show Realtime Latency
Enable the display of the latency in the status bar, for more information see chapter "Real time latency" on page 275.

The **Plot** options are used to control some plot window settings.

Show toolbar when opening new plot window
Enable the toolbar automatically in a new plot window.

Minimum analog & force real time plot update
Set the update time for the analog and force plots in the real time mode. Use this

setting to change how much of the analog and force data that is shown in a plot. A shorter time will give a higher resolution, i.e. more samples are shown per second. A longer time can be used if you want to see a longer course of events, but then less samples are shown per second.

Note: Making the plot window wider also increases the number of analog samples in the plot.

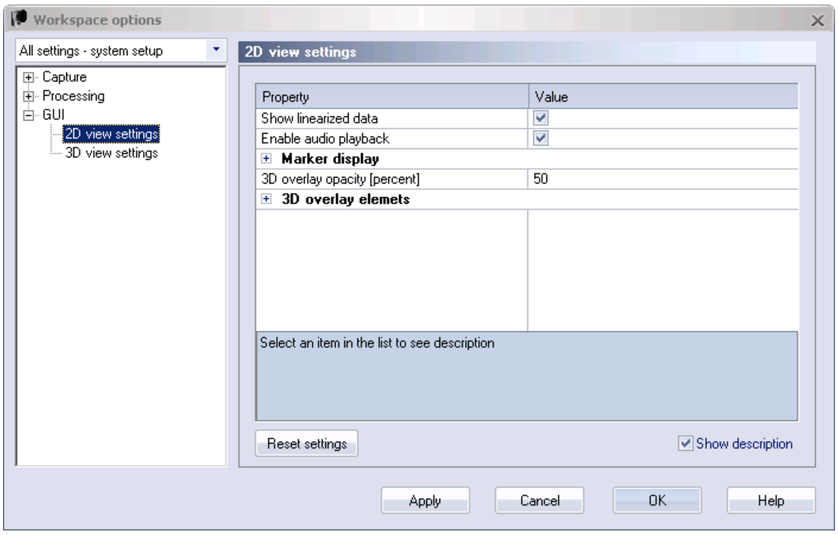
The **Measurement file close** options are used to control how QTM closes files automatically.

When opening another file
The current file is closed when opening another file (**Open** in the **File** menu).

When beginning a new measurement
The current file is closed when starting a new measurement (**New** in the **File** menu).

Use the **Reset hidden message boxes** button to display all message boxes that have been hidden. E.g. the message box about the force plate position after a calibration.

2D view settings



The **2D view settings** page contains settings for objects that is shown in the 2D view window. The settings are saved with the workspace and are applied both in the RT/preview and on a opened file.

The following settings can be changed use the **Reset settings** button if you want to reset all of the settings to default.

Show linearized data
Toggle whether linearization is applied to the marker and video data in the 2D views. The unlinearized data is the original data, while the linearized data is the one that is used for 3D tracking. Therefore it is best to use linearized data when looking at the 3D overlay, because otherwise the 3D positions will not match the 2D positions.

Enable audio playback
Enable audio playback for captured video from DV-cam and webcam.

Marker display

All markers have the same size
Select whether the 2D markers are displayed with their actual size or with the same size.

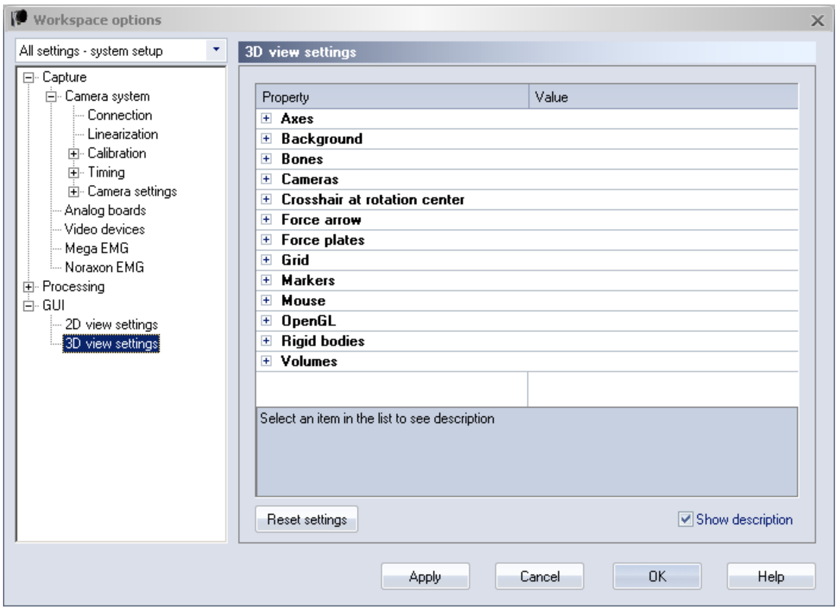
Marker size
Select the marker size when **All markers have the same size** is set to **Yes**. For Oqus this value is in subpixels and for ProReflex it is in 1/1000 of diagonal FOV.

3D overlay opacity [percent]
Set the opacity for the 3D overlay. The default is 50. 0 makes the overlay invisible and 100 is almost no opacity, which means that the 2D data or video is hardly visible.

3D overlay elements
Select the 3D elements that will be displayed in 3D overlay. The following elements can be controlled in the 3D overlay.

Markers, **Marker traces**, **Bones**, **Grid**, **Axes**, **Cameras**, **Force plates**, **Force arrow**, **Force trace**, **Rigid bodies**, **Volumes**, **Bounding box**

3D view settings



The **3D view settings** page contains settings for objects that is shown in the 3D view window and also how to operate the 3D view. The settings are saved with the workspace and are applied both in the RT/preview and on a opened file.

The following settings can be changed use the **Reset settings** button if you want to reset all of the settings to default. Only the more complex settings are explained below, for explanation of the other settings please check the description in the **Project options** dialog.

Axes
Display settings for the global axes in the 3D view.

Show axes, **Length of the axis (mm)**

Background
Display setting for the background in the 3D view.

Background color

Bones
Display settings for the bones in the 3D view.

Show bones, **Bone thickness (mm)**, **Bone color**

Cameras
Display settings for the cameras in the 3D view.

Show cameras, **Show camera IDs**

Crosshair at rotation center
Display settings for the crosshair shown when zooming and translating in the 3D view.

Show crosshair, **Crosshair color**, **Crosshair size (pixels)**

Force arrow
Display settings for the force arrow in the 3D view.

Show force arrow, **Arrow color**, **Scale factor (mm/N)**, **Show force trace**, **Force trace color**
The **Scale factor** option sets the size of the force arrow in relation to the force in N.
Activate the force trace, also called force butterfly or Pedotti diagram, with the **Show force trace** option. The force trace is shown for the active measurement range.

Force plates
Display settings for the force plate in the 3D view.

Show force plates, **Force plate color**,

Grid
Display settings for the grid in the 3D view.

Show grid, **Show grid measure**, **Grid color**, **Automatic size**, **Grid length (m)**, **Grid width (m)**, **Vertical offset (mm)**, **Distance between each gridline (mm)**
The **Automatic size** option will make the grid approximately the same size as the lab area, i.e. the size is set by the camera positions.

Markers
Display settings for the markers in the 3D view.

Use global marker size setting, **Marker size (mm)**, **Show marker labels**, **Show marker traces**, **Traces line style**, **Show trajectory count**
The **Use global marker size setting** option will decide whether to use the same marker size on all files or individual settings for each file. Disable the option to use the marker size that was used when a file was saved.

The **Show trajectory count** option will display the number of trajectories in the current frame in the bottom right corner of the 3D view.

Mouse

Mouse behavior, **Default zoom method (The other: Press Ctrl)**
The **Mouse behavior** can be set to: **Qualisys standard**, **Visual3D emulator** and **MotionBuilder emulator**. The two emulators change the mouse behavior to the same as in the respective program.
Note: When using the **MotionBuilder emulator** the following two mouse behaviors are deactivated because of a conflict: select trajectories in an area (Shift + Left-click + drag) and the zoom alternative (ctrl + zoom).

OpenGL

OpenGL format selection mode
The default values will give the best quality of graphics. However if there are any problems with the graphics first try to disable the anti-aliasing, then try one of the other modes: **Use options below** or **Use explicit index**.

Use full scene anti-aliasing if available
Anti-aliasing smooths the edges of the graphics, but it can also slow down the playback. This feature is not available on graphic boards where the driver doesn't support OpenGL 2.0.

OpenGL double-buffering (only available for **Use options Below**)
The default setting is to **Use double buffering**. But to reduce the workload on the graphic board you can try **Use front buffer only** or **Use single buffer only**.

Use HW acceleration (only available for **Use options Below**)
Disable the acceleration if the graphic board has no built-in memory.

OpenGL format number (only avialable for **Use explicit index**)
Set the index for the OpenGL format, however some indexes might not work at all. To see a list of the available formats click on the ... button.

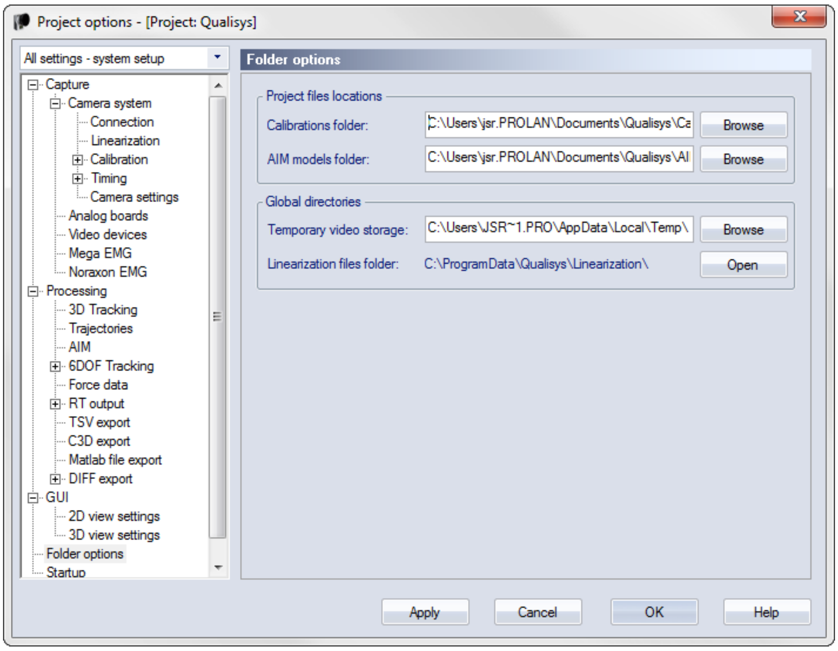
Rigid bodies
Display settings for the rigid bodies in the 3D view.

Show rigid bodies, **Show coordinate system axes**, **Length of the coordinate system axes (mm)**, **Show rigid body points as markers**, **Show rigid body labels**
Activate the display of the rigid body points in the 6DOF definition, with the **Show rigid body points as markers** option. The markers are calculated from the 6DOF definition and therefore they will be displayed even if the corresponding marker is hidden.

Volumes
Display settings for the display of volumes in the 3D view, for more information see chapter "Volumes in 3D views" on page 57 and "Camera view cones in 3D views" on page 56.

Show covered volume, **Cut covered volume at floor level**, **Cameras required to consider volume covered**, **Show calibrated volume**, **Enable camera view cones**, **Minimum visible marker size [mm]**, **Show bounding box**

Folder options



The **Folder options** page contains the settings for the locations of the following file types saved by QTM.

The **Project files locations** are different for each project.

Calibration folder
The **Calibrations folder** is by default set to a folder called Calibrations in the Project folder. It can be changed to another folder with the **Browse** option, for example if you want all of the calibrations in different projects to be saved in the same folder. However, then you must change it manually in each project.

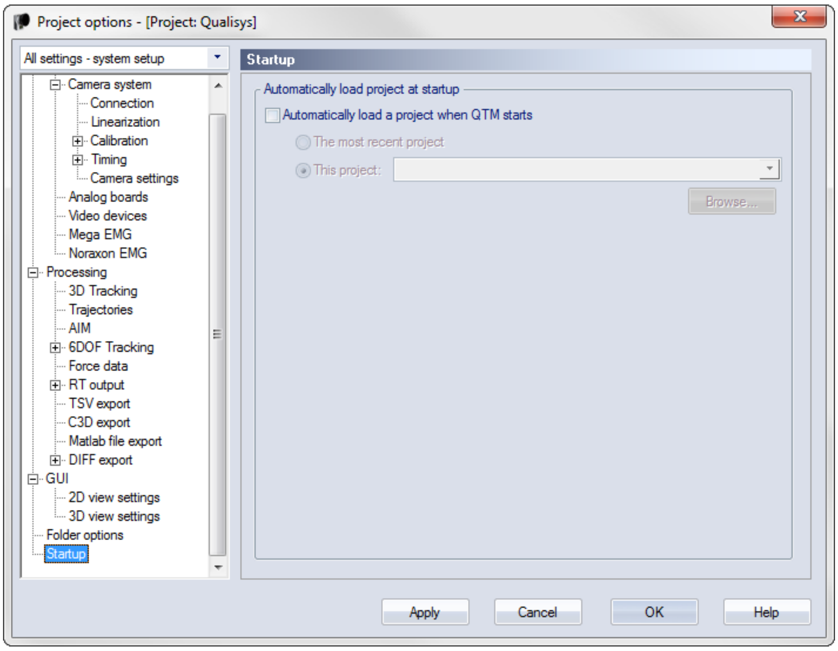
AIM models folder
The **AIM models folder** is by default set a folder called AIM models in the Project folder. It can be changed to another folder with the **Browse** option, for example if you want all of the AIM models in different projects to be saved in the same folder. However, then you must change it manually in each project.

The **Global directories** are used by all projects.

Temporary video storage
The **Temporary video storage** is used for storage of the temporary video files of Oqus and Point grey cameras. It can be changed to another folder with the **Browse** option, which can be useful if the actual file will be stored on another hard-drive than the temporary file.

Linearization files folder
The camera linearization files are stored in a folder that can be accessed by all users of the computer.

Startup



The options on the **Startup** page changes how projects are loaded when QTM starts. The default is that no project is loaded automatically and that you have to select a project from the **Open project** dialog, see chapter “Switch project” on page 11.

However, if you want to load a project automatically, then activate the option **Automatically load a project when QTM starts** and select one of the following two options.

The most recent project
Loads the project that was last opened in QTM.

This project
Loads a selected project when QTM starts. Select the project with the **Browse** option.

Making a measurement

Connection of the camera system

Camera system connection

Before the first use of the camera system it must be connected to QTM. Make sure that the camera system hardware has been correctly installed, see chapter “Setting up the system (Oqus)” on page 329 and “Setting up the system (ProReflex” on page 347.

The connection of the camera system to the QTM software is performed on the **Connection** page in the **Project options** dialog, see chapter “Connection” on page 111. The connection can be done automatically or manually. The automatic connection should be used as long as there are no problems with the camera communication. When the connection has been made, the camera system can be used as long as the properties are not changed.

Note: When locating the system QTM will detect automatically if the camera system has old firmware. The firmware must then be updated before the system can be used. For more information see chapter “Firmware update when locating system” on page 353.

Outline of how to locate the camera system

The steps below are just an outline of what should be done to automatically connect the camera system to QTM.

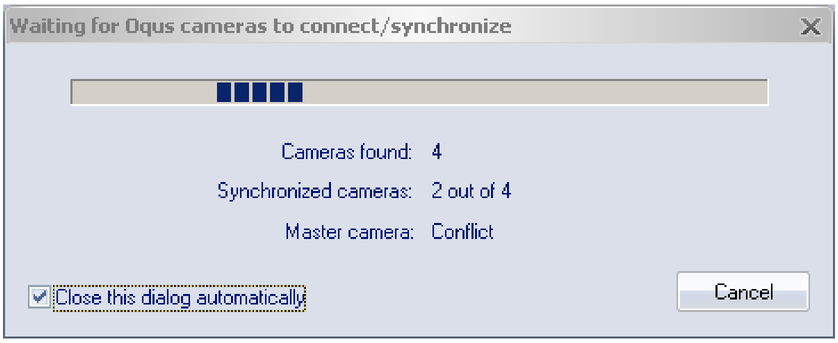
Follow these steps to connect the camera system to QTM:

1. Switch on the camera system, wait for the cameras to start up properly and start QTM.
2. Open the **Project options** dialog and go to the **Connection** page.
3. Click **Locate system**.
4. Click **Start**.
5. Choose the camera system and click **OK**.

Note: If any problems with the connection process occur, check the options on the **Connection** page in **Project options** dialog again. If that does not help, check the troubleshooting list in chapter “Troubleshooting connection” on page 387.

Waiting for Oqus synchronization after new file

QTM will wait for Oqus cameras to be synchronized when opening a new file with **New** in the **File** menu. The dialog below is shown until all of the Oqus cameras are synchronized.



The dialog contains the following information.

Cameras found
Number of cameras found by QTM.

Synchronized cameras
Number of synchronized cameras.

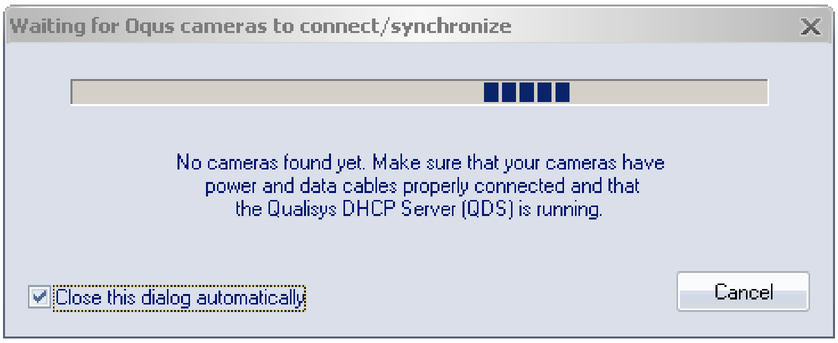
Master camera
Status for the master camera.

Uncheck the **Close this dialog automatically** checkbox if you do not want the dialog to close.

QTM will wait until the following conditions are fulfilled.

1. For the number of cameras there are two different conditions depending on whether the system has been located on the **Connection** page in the **Project options** dialog.
    a. If the system has been located, all cameras that was in the system must be found.
    b. If the system has not been located, QTM will only wait until all of the cameras found during the waiting time is synchronized. Then it is good idea to uncheck **Close this dialog automatically** so that you can check manually that all cameras have really been found.
2. All cameras must be synchronized.
3. That there is only one master camera.

A special case is if you start a new file before any camera has started. Then if a system has been located before, QTM will start waiting for the cameras to startup and synchronize, see dialog below.



Linearization of the cameras

Camera linearization

A camera transmits coordinates in two dimensions for each marker's center point in the field of view of the camera. The locations of the center points reflect the optical view of the sensor chip in the camera. Since this cannot be done without distortion, the QTM software must have information about the distortion of each camera in order to get correct 3D data. This is achieved by an adjustment of the camera data with a linearization procedure in QTM, which is then stored as a linearization file.

Most of the distortion occurs because of the lens type and the lens mounting alignment in reference to the mounting of the sensor. For short focal length lenses, lens distortion is also a significant factor that needs correction.

The linearization procedure uses an individual correction file for each camera, which is supplied with each individual camera. For Oqus the files are stored in the camera memory, but for ProReflex they are supplied on a CD. The linearization files of the current system are specified on the **Linearization** page in the **Project options** dialog, see chapter "Linearization" on page 116. On the **Linearization** page you can also deactivate cameras in the camera system.

Note: If any of the factors which influence the linearization is changed for a particular camera (e.g. lens changed or lens position on lens mount changed) a new linearization correction file must be created. The camera must then be shipped to an authorized Qualisys agent or distribution center for re-linearization.

Linearize a camera

The following chapters describe how to linearize an Oqus or a ProReflex MCU with the QCC method. These instructions must be followed to ensure that the linearization is correct.

Concept of QCC linearization

The concept of QCC linearization is to capture frames with the plate in as many different positions and orientations as possible. Therefore there are several constraints made to the measurement so that the correct frames are used in the linearization.

1. The 2D image is divided into 7 times 6 squares.
2. Each square is divided into four angle cells, which is visualized as triangles. In each angle cell you have to collect at least 20 markers.
3. The size of the markers must be at least 7 for ProReflex and 200 for Oqus.
4. The plate must have at least one vanishing line, i.e. the angle of the outer lines of the square must be large enough.
5. The angle of the plate to the plane of the sensor must be more than 10 degrees.
6. The angle between a corner marker and the line of sight to the camera must be less than 40 degrees.
7. At least 22 of the markers must be visible in the frame.

8. There must be no more than 30 markers in the frame, i.e. no other reflexes than the markers on the plate.

These constraints are implemented in the linearization process so you cannot collect frames that are outside of the limits, which means that you can move the plate without thinking to much about the limits.

Linearization instructions

Follow these instructions to linearize.

1. Start QTM.
2. Place the camera on a tripod. It is a good idea to place the camera side by side with the computer screen. Because it is easier to linearize if you have the screen right in front of you when you move the plate.
3. Connect the camera to the computer and open a new file. You can have several cameras connected while linearizing, but you can only linearize one camera at a time.
4. The Real time frequency is changed to 60 Hz automatically when you start the linearization.
5. **Oqus 1-series:** Set the aperture to 4.

   Oqus 3-series and Oqus 5-series: Set the aperture to 2.8.

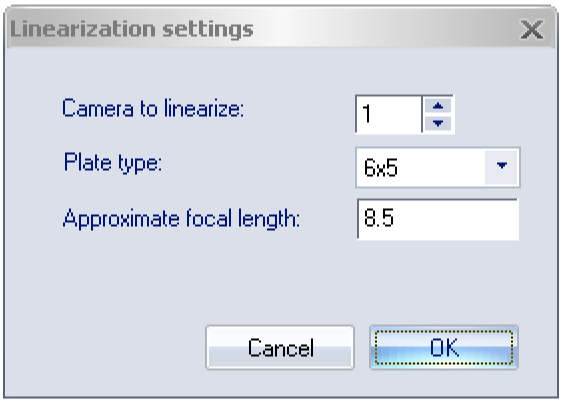
   3+-series and 4-series: Set the aperture to 2.8, but you can use higher numbers up to about 5.6 to make it easier to focus.

   ProReflex: Set the aperture to a little bit more than 8 for a 6 mm lens. For the longer lenses it can be between 8 and 4.

   If there are no numbers on the lens, hold the linearization plate at about 0.7 m for a 6 mm lens and close the aperture until there are no reflexes when you move the plate. For other lenses you can calculate the measurement distance from the table below.
6. **Oqus 1-series:** Set the focus at about 1.5 m for 8.5 mm lenses. For the best result place the linearization plate at about that distance and set the focus. If you have a another focal length you multiply with that factor. For example a 16 mm lens should have the focus at about 3 m.

   Oqus 3-series, 3+-series, 4-series and Oqus 5-series: Set the focus at about 3 m for 25 mm lenses. For the best result place the linearization plate at about that distance and set the focus. If you have a longer focal length you multiply with that factor. For example a 50 mm lens should have the focus at about 6 m.

   ProReflex: Set the focus to infinity or very close to infinity. The focus should be somewhere at the end of the longer measurement distance, e.g. for 6 mm lenses it could be around 2.5 m, see table below.
7. **IMPORTANT:** You might need to take off rings or other shiny objects when you perform the linearization. For Oqus cameras change the **Marker threshold** and **Exposure time** so that there are no extra reflections when you stand in front of the camera.
8. Click the **Linearization** button or click **Linearize camera** in the **Capture** menu. The following dialog will appear:



IMPORTANT: During the rest of the linearization process you must **not** touch the lens. However the camera can be moved slightly if it is absolutely necessary.

9. Select the number of the camera that you want to linearize with **Camera to linearize**.

10. You do not need to change the **Plate type**. However if you want to check the type, it depends on the number of markers.

11. **Oqus:** The **Lens manufacturer** setting is not available for Oqus.

    ProReflex: Select **Goyo** as the **Lens manufacturer**. If you are not sure that you have another lens.

12. Enter the focal length of the current lens in **Approximate focal length**. The measurement distances for the linearization will depend on the focal length and the camera type according to the tables below.

    ProReflex: The standard lens is 6 mm. If you do not know what the focal length is you can check the second letter in the name of the old linearization file, see table below.

Focal length (mm)	Letter in the Lin-file	Short distance (m)	Long distance (m)
4.5	B	0.3-0.7	0.8-1.2
6	D	0.4-0.9	1.1-1.6
8	E	0.5-1.2	1.5-2.1
12	F	0.8-1.8	2.2-3.2
16	H	1.1-2.4	2.9-4.3
25	I	1.7-3.8	4.6-6.7
35	L	2.3-5.3	6.4-9.3
50	J	3.3-7.5	9.2-13.3

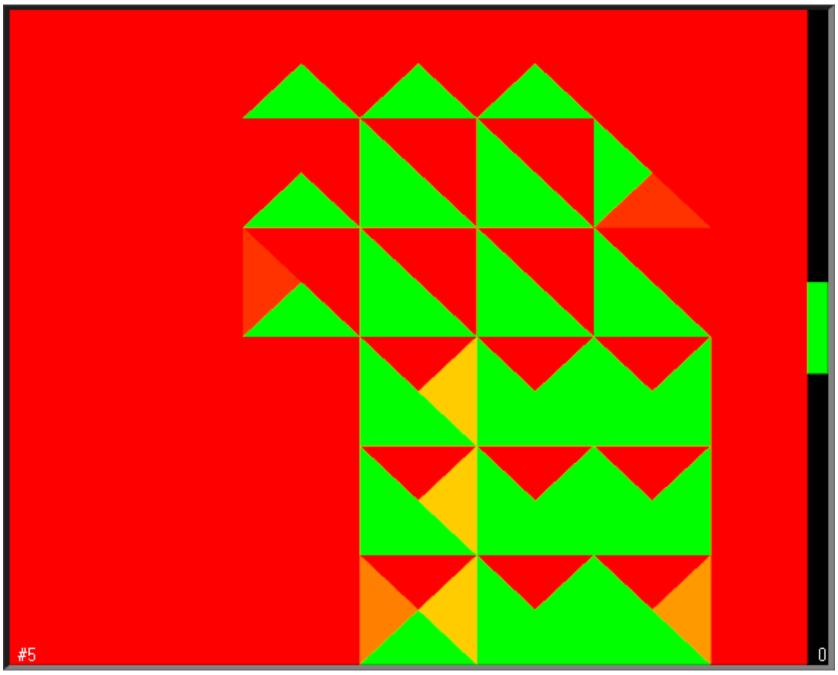
Oqus 1-series: The standard lens is 8.5 mm.

Focal length (mm)	Short distance (m)	Long distance (m)
6	0.3-0.7	0.8-1.3
8	0.4-1.0	1.2-1.8
12	0.6-1.4	1.7-2.5
16	0.8-1.9	2.3-3.4

Oqus 3-series, 3+-series, 4-series and Oqus 5-series: The standard lens is 25 mm (16 mm for 4-series).

Focal length (mm)	Short distance (m)	Long distance (m)
16	1-1.4	1.8-2.2
25	1.6-2.2	2.8-3.5
50	3.2-4.4	5.6-7.0
75	4.8-6.6	8.4-10.5

13. When you have selected the focal length, it is time to start the actual linearization process. When you click on OK the 2D view will turn red. In the image below the linearization process has started.



14. The following instructions are shown in QTM to help with the linearization:
    a. There is a vertical bar to the right in the 2D view. The white diamond shows the current distance of the plate to the camera and the green area is where you must be to linearize.
    b. An arrow pointing up means that you must move closer to be at the correct distance.
    c. An arrow pointing down means that you must move further away.
    d. When the markers are too small, the error **Too small markers** are shown at the bottom left corner. If this happens all the time move closer or start over with a more open aperture.
    e. The 2D view is divided into 7 times 6 squares and each square is divided into four triangles. Angle the plate towards direction of the triangle to collect data in that angle.
    Note: For Oqus cameras with focal lengths shorter than 24 mm and underwater cameras there are only 5 times 4 squares.
15. The linearization process can be summarized in these steps for the two distances.
    a. First three general instructions for both distances.

i. Start with the corners and try to move the plate close to the edge of the field of view. Then you can take the squares that have not turned green.

ii. The fastest way to collect the frames is to just angle the plate vertically and horizontally. When all of the triangles have turned green you have finished a distance.

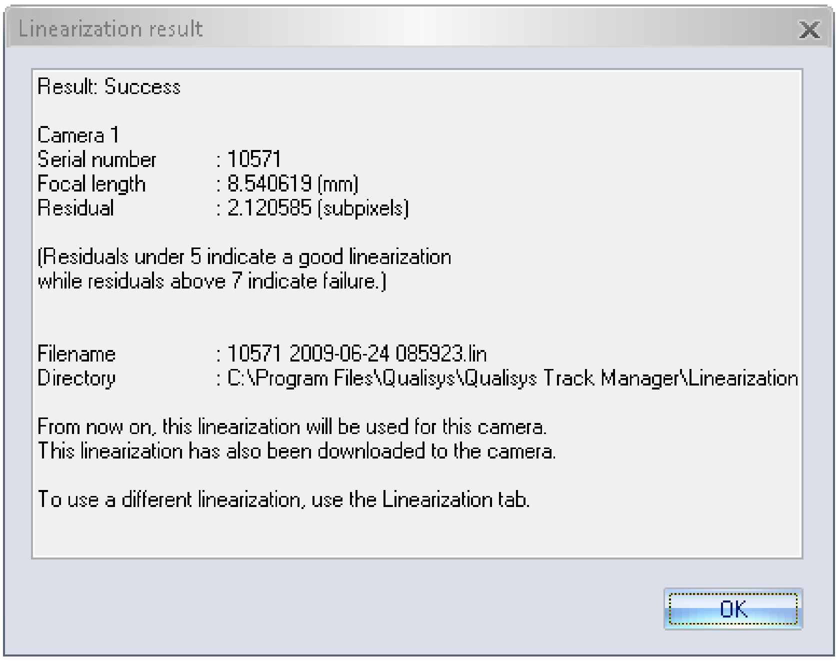
iii. Make sure that you do not hold any fingers so that they can block a marker.

b. First move the linearization plate within the short distance. Try to stand as far away from the upper limit as possible, but make sure that the whole plate can be seen. E.g. for a ProReflex with 6 mm lens the distance that you can use is between 0.7 to 0.9 m.

IMPORTANT: (ProReflex) Make sure that the camera can calculate all of the markers in the frame. If the bottom markers disappear now and then you must close the aperture a bit more or move further away.

c. Move back to the longer distance.
Oqus: Sometimes the markers are too small at the longer distance. If that happens restart the linearization with a lower Marker threshold or a lower Exposure time.
ProReflex: It is quite easy for the markers to become too small at the longer distance. If the markers are too small most of the time, try moving as close as possible or restart the linearization and open the aperture more.

16. When you are finished with the second distance QTM will calculate the linearization parameters. The dialog that then appears tells you if the linearization passed.



IMPORTANT: The two significant results are the **Focal length** and the **Residual**. Check that the **Focal length** is correct and that the **Residual** is smaller than 5. If the residual is larger than 5, check that the focus was correct and try again if it was not. If you cannot get a residual below 5 please contact Qualisys AB for service of the camera. This does not apply if you have a 4.5 mm lens on a ProReflex, then the residual should instead be smaller than 6.

17. When you click OK in the dialog the linearization is stored on the **Linearization** page in **Project options**. For Oqus cameras the linearization file is also stored in the camera.

    Three files are saved on the computer in the folder **Linearization**. The first two files are named with the serial number followed by the date and time. **ProReflex:** For ProReflex the serial number is followed by a four letter code, e.g."3137SDGB 2006-05-03 102222".

    The three files are:

    .lin - The linearization file that is used by QTM.

    .stat - This file contains the settings of the linearization and also some statistics.

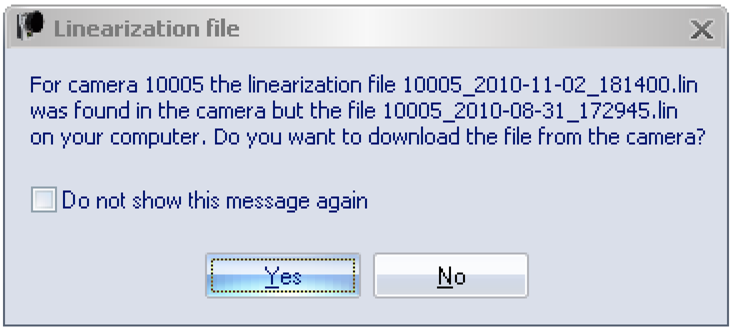
    .qcl - This file contains the actual linearization measurement. This is only named with date and time.

    The last two files can be sent to Qualisys AB if you have any problem with the linearization. Otherwise you do not need to look at those files.

18. Delete the old files from the computer so that you do not use them by mistake, you can save a copy somewhere else if you want to. If you have saved workspaces in QTM it is a good idea to load them, change the lin files and resave the workspace.

Linearization file warning

The linearization file in the camera is compared to the one used in QTM every time that you start a new measurement. If the files do not match the warning below is displayed.



If you want to download the file from the camera click **Yes**, which you should probably do if the file in the camera is more recent than the one in QTM. Otherwise click **No** and check the box **Do not show this message again** to not see the warning for this file again.

Calibration of the camera system

Introduction to calibration

The QTM software must have information about the orientation and position of each camera in order to track and perform calculations on the 2D data into 3D data. The calibration is done by a well-defined measurement procedure in QTM. Calibration is, however, not needed for a 2D recording with only one camera.

There are three methods that can be used to calibrate a camera system: Wand calibration, Frame calibration and Fixed camera calibration. The methods are described in chapters below.

The following items are important to think about before the calibration:

- Before the calibration make sure that the calibration settings are correct on the **Calibration** page in the **Project options** dialog, see chapter "Calibration" on page 119. The focus and aperture must also be set before the calibration.
- When using Wand calibration or Frame calibration it is important that the camera system have been placed correctly to achieve a high-quality calibration, see chapter "Camera positioning" on page 250. For Fixed camera calibration it is equally important with the camera positions but in this case the cameras are fixed and must be installed correctly before the calibration.
- It is recommended that the motion capture system is calibrated before each measurement session to make sure that the captured data has high quality. Check regularly that the calibration is OK during long measurement sessions. There is an automatic check if the cameras has moved, which is activated on the **Calibration Quality** page in the **Project options** dialog, see also chapter "Calibration quality warning" on page 240.
- For ProReflex the calibration will run in different modes depending on the capture rate so that the light conditions will be the same, see chapter "Camera system" on page 108. But you can calibrate in one mode and capture in another by lowering the capture rate before the calibration.

A calibration starts with the **Calibration** dialog, see chapter "Calibration dialog" on next page. Click on the **Calibration** icon to open the dialog.

Note: Each calibration is saved in a QCA file in the **Calibrations** folder in the project folder, see chapter "Project folder" on page 12. The file name contains the date and time of the calibration.

Important: Each time a camera in the system is moved (even the slightest) a calibration must be performed before high-quality data can be captured again. However, some errors with a calibration can be fixed with reprocessing, see chapter "Reprocessing a file" on page 281.

Outline of how to calibrate (Wand calibration)

The steps below are just an outline of what should be done to calibrate the camera system with Wand calibration method.

Follow these steps to calibrate the camera system:

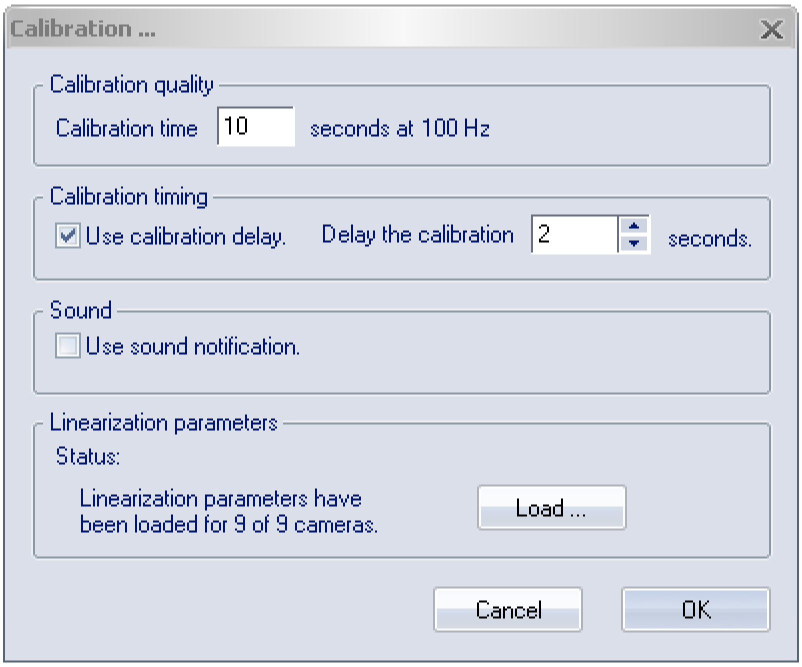
1. Switch on the camera system and start QTM.

2. Open a new file by clicking the **New file** icon .
3. Place the L-shaped reference structure in the measurement volume, see chapter “Wand calibration method” on page 241.
4. Set the settings on the **Calibration** page in the **Project options** dialog, see chapter “Calibration” on page 119.
5. Click **OK**.
6. Click the **Calibration** icon and set the settings in the **Calibration** dialog, see chapter “Calibration dialog” below.
7. Click **OK**.
8. Move the calibration wand in the measurement volume, see chapter “Wand calibration method” on page 241.
9. Check the **Calibration results** and click **OK**.
   Note: If you have force plates there will be a warning reminding you of measuring the force plate position again. Since it has most probably changed some with the new calibration.

Note: If any problems with the calibration process occur, check the settings on the **Calibration** page in the **Project options** dialog and in the**Calibration** dialog. If that does not help, check the troubleshooting list in chapter “Troubleshooting calibration” on page 389.

Calibration dialog

A calibration process is started with the **Calibration** dialog. It is opened by clicking either **Calibrate** on the **Capture** menu or the **Calibration** icon .



Calibration quality

The frequency in the calibration is fixed, the default value is 100 Hz. This frequency is set to ensure a good number of frames in the calibration and to give a good visualization when the calibration is tracked. The frequency will be lowered if the camera system cannot measure at 100 Hz with the current settings.

50 Hz - If an Oqus camera is set to a exposure time longer than 1 ms.

10 Hz - For underwater Oqus cameras.

Set the **Calibration time** to a time longer than 10 s. QTM will use up to the value set with the **Maximum number of frames used as calibration input** setting on the **Calibration** page in the **Project options** dialog. The default value is 1000 frames used in the calibration. The frames used in the calibration process will be distributed evenly when the calibration has more frames than specified with the setting.

To achieve the best possible calibration in a Wand calibration, it is necessary to capture frames of the calibration wand in as many and as different positions as possible within the measurement volume. The time should therefore be long enough so that the person, who moves the wand, has time to move it to a lot of different positions.

For Frame and Fixed camera calibration the length of the calibration is not especially important, since the data of the frames is averaged. Therefore the minimum calibration time of 10 seconds can be used.

Calibration timing

Under the **Calibration Timing** heading a delay (in seconds) of the start of the calibration can be specified.

Sound

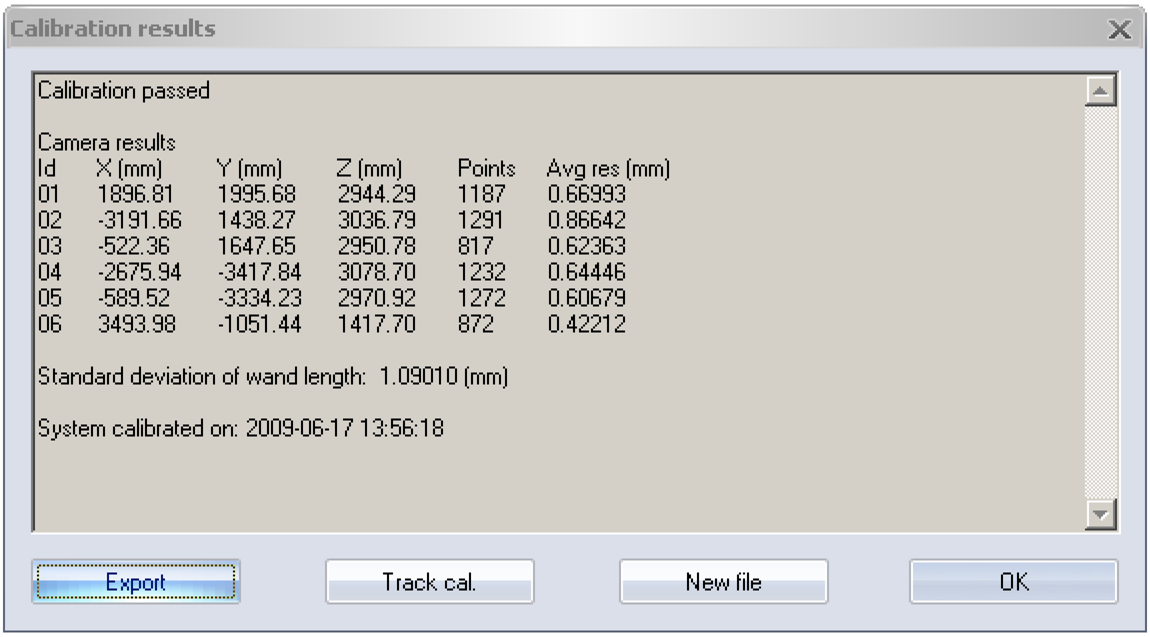
Use the **Sound option** to choose whether to play a sound when the calibration starts and ends.

Linearization parameters

Under the **Linearization parameters** heading the status of the linearization parameters is shown. The linearization files must be specified. Otherwise you cannot start the calibration. Click **Load** if the files have not been specified.

Calibration results

The **Calibration results** dialog is shown after a calibration is completed. It displays if the calibration passed and the calibration quality results. The error messages of a failed calibration are described in the chapter “Calibration failed” on page 239.



There are four buttons at the bottom of the dialog, clicking them have the following effects.

OK
Close the dialog and the calibration file.

New file
Close the dialog and the calibration file and open a new capture file in preview mode.

Track cal.
Track the calibration file and close the dialog.

Export
Export the calibration results to a xml file. The exported file also includes the rotation matrix for the cameras. The default folder to save in is the **Calibrations** folder in the project folder, see chapter "Project folder" on page 12.

The current calibration can be viewed on the **Current calibration** page in the **Project options** dialog. The page can be opened by double-clicking the calibration status icon in the bottom right corner of the **Status** bar.

Note: In a capture file the calibration results can be examined by clicking the **Reprocessing** icon and then opening the **Calibration** page.

Quality results

The quality results under the **Camera results** heading are camera specific. For each camera ID there are the following five results:

X (mm), Y (mm) and Z (mm)
The distance (in mm) between the origin of the coordinate system of the motion capture to the optical center of the camera. The distances are respectively in the X, Y and Z direction.

Points
Number of points used in the calculation of the distance above. The number should be as many as possible, but without large differences between the cameras. The maximum number of points for a Wand calibration depends on the calibration time and the number of frames used in the calibration. If the camera has more than

500 points it is usually enough for a normal measurement volume. For the other methods it depends on the number of markers seen by the camera.

Avg. res. (mm)
The average residual (in mm) for the points above. The residual of the cameras should be similar in size and as low as possible. Depending on the measurement volume the average residual can differ between 0.5 to 1.5.

Note: If the camera result says **Unused camera**, then the camera has been deactivated on the **Linearization** page in the **Project options** dialog. That camera cannot be used in measurements, unless the calibration is reprocessed, see chapter “Reprocess calibration” on page 244.

For a Wand calibration there are also a general quality results:

Standard deviation of wand length
The standard deviation (in mm) of the wand length in the calibration.

Finally the calibration time when the calibration was performed is displayed at the end.

Track cal.

With **Track cal.** the calibration is tracked and opened in a **3D view** window. For a Wand calibration the movements of the wand is shown and the measurement volume can be confirmed. For the other two methods the positions of the markers can be confirmed in the 3D view.

Note: If the calibration is opened, the window must be closed before a new capture can be started.

Calibration failed

If the calibration fails the calibration result will say **Calibration failed** and an error message is displayed after each camera ID.

The error messages are as follows:

General calibration failure
Something is wrong in the calibration. Check the calibration settings.

Wand calibration errors:

Couldn't find the fourth marker of the L-frame
The marker on the short leg of the L could not be found. Check the reference structure and that all four markers are visible in the preview window of the camera.

Couldn't find the three markers in line on the L-frame
The three markers in line on the long leg of the L could not be found. Check the reference structure and that all four markers are visible in the preview window of the camera.

One or more of the markers on the L-frame were unstable
The reference structure or the camera has moved during the calibration. Make a new calibration.

Could not initialise camera
There is some general failure so that the camera cannot calculate where the L-frame is. One possible reason is that it is the wrong linearization file.

Frame and Fixed camera calibration errors:

Less than 75% of the frames had the correct no of markers in stable positions
Some of the markers have been hidden or moved during the calibration. Make a new calibration.

Wrong number of markers seen by the camera in the first frame
The definition of markers seen by each camera does not match the markers in the first frame. Check that the calibration settings on the **Calibration** page in the **Project options** dialog are correct.

No markers defined to be seen by this camera
There are no markers specified in the calibration settings for this camera. Check which markers that are seen and enter them on the **Calibration** page in the **Project options** dialog.

Calibration quality warning

The calibration quality is shown visually with the calibration status icon in the right corner of the status bar. QTM checks both the residual of the last file and time since the last calibration. The checks are activated on the **Calibration quality** page in the **Project options** dialog.

The default value for the residual check is 3 mm and it also checks if too few 2D markers of the camera is used in the 3D tracker. For the time the values are 5 h and 12 h, so that you are reminded to calibrate the system each day.

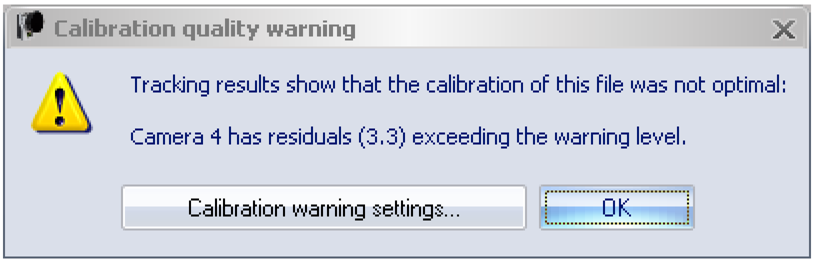
 The calibration is new and ok.

 The calibration is older than the first time limit or you have had a warning from the residual check after the last measurement.

 The calibration is obsolete, i.e. it has passed the second time limit. It is recommended to calibrated again, however it can still be used.

 The system is not calibrated.

If residual check is exceeded when a file is tracked after a measurement, there is a warning describing the error. For example if one of the cameras exceed the limit you will get the warning below.



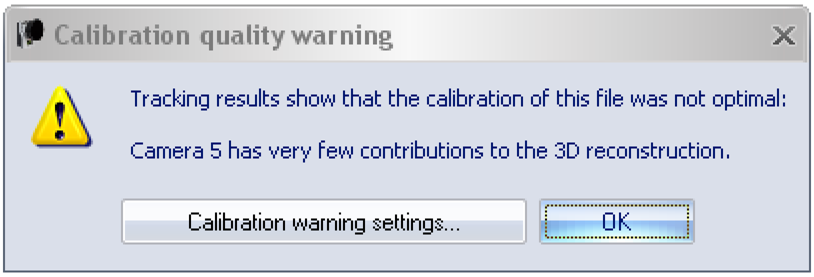
For residuals that are just slightly over the limit the data will still be ok, especially if you have more than 2 cameras that can see each marker. However it is recommended that you check the setup to see if it can be improved and then calibrate the system.

- First check that the markers 2D size of the markers are not much smaller than in the other cameras. This can cause a higher residual than normal.
- Then you should check if the camera might have moved. For example if a screw is not tight enough in the mounting the camera might move slightly by its own weight causing a residual that slowly increases. This can be controlled by making a measurement of just static markers 1-2 hours after the

calibration. Then if you get the error message it is very likely that the camera has moved slightly.

If the warning instead is about the camera having too few contributions, then you get the warning below. In this case the data is not used in the 3D tracking so the 3D data you have has not been degraded. However it is recommended to check the 2D data to see what the reason is and then calibrate the system.

- Turn on the 3D overlay and check if the 2D markers match the 3D markers. It could be that the camera has moved so that it no longer contributes to the 3D data.
- Then check if there are a lot of extra markers. The warning can also be caused by extra markers that do not result in 3D data.



Wand calibration method

The Wand calibration method uses a calibration kit that consists of two parts: an L-shaped reference structure and a calibration wand. The L-shaped reference structure is actually a mirrored L.

Place the L-shaped reference structure so that the desired coordinate system of the motion capture is obtained. It is best if all cameras in the system can see all markers on the reference structure. This is not an unconditional requirement but it gives the highest accuracy. If some cameras cannot see the reference structure the Extended calibration method is used automatically, see chapter “Extended calibration” on next page.

The calibration wand is moved inside the measurement volume in all three directions. This is to assure that all axes are properly scaled. The calibration algorithms will extract each camera's position and orientation by evaluating the camera's view of the wand during the calibration. For more information on how to move the wand see chapter “Calibration tips” below and “How to move the wand” on next page.

Note: A suitable setup to assure a correct calibration of all cameras is to use 20-30 seconds for the calibration. It is, however, very dependent on the size of the measurement volume. You need to specify enough time to cover the entire volume with the wand.

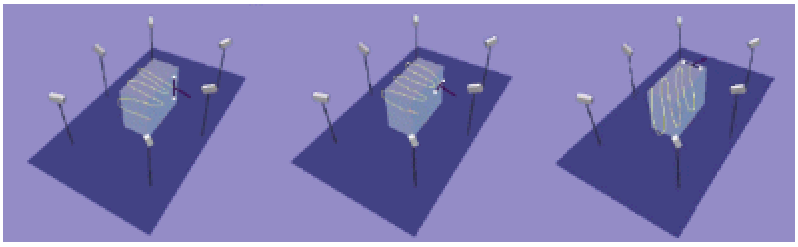
Calibration tips

During the calibration, the wand is preferably moved in the measurement volume in a way that allows all cameras to see the wand in as many orientations as possible. This way the cameras will be properly calibrated in all directions.

To assure that all cameras can see the wand as much as possible during the calibration, make sure that the cameras are not blocked by the person moving the wand around in the measurement volume. This ensures that no camera will be blocked for a longer time period.

How to move the wand

One suggestion on how to move the wand, is to move it in one direction at a time. Start by holding the wand positioned in the Z direction, i.e. the straight line between the two wand markers should be parallel to the Z axis. Move the wand in the entire measurement volume. It is important to fill the entire measurement volume with calibration points. Make sure that both the lower and upper parts of the volume are covered. Repeat the same procedure with the wand positioned in the X and Y direction. It is particularly important to collect points where there will be many markers during the motion capture.



Note: In this picture the reference structure is not indicated to make the picture more distinct. The reference structure must of course always be present during the calibration. The box in the figure represents the measurement volume.

Another wand movement procedure is described below.

1. Move the wand back and forth as close to the bottom as possible without moving the reference structure. The wand should be parallel to the floor, i.e. positioned in the X or Y direction.

2. Collect calibration points in the entire measurement volume. It might be a good idea to rotate the wand in the volume, particularly if the volume is large. This method gives many calibration points during a short time period and the wand is detected in many directions without any large effort. Do not rotate at a very high speed.

Note: There are many other ways to move the wand and the optimal moving method varies depending on the application, i.e. you have to find out the best method for your application. It is recommended that the moving method is systematic and easy to repeat.

Extended calibration

To enable a larger measurement volume QTM uses a method called Extended calibration. The Extended calibration is used automatically as soon as a camera cannot see the L-shaped reference structure, but has an overlap with other cameras.

In Extended calibration there is only some of the cameras that can see the L-shaped reference structure. The other cameras are calibrated by the overlap with the reference cameras field of view. The result is a larger volume but with reduced 3D data accuracy compared with when all cameras can see the reference structure.

To perform an Extended calibration, make sure that at least two cameras can see all markers of the reference structure. Then to extend the measurement volume, the following three conditions must be fulfilled:

1. All parts of the measurement volume must be seen by at least two of the cameras. Since it is impossible to calculate 3D coordinates in parts of the volume where only one camera can see the markers.

2. The whole measurement volume covered by the cameras must be connected. This means that a marker must be able to move through the whole volume without passing any volume that less than two cameras cover.
3. In volumes where only two cameras overlap, the size of the volume must be large enough to fit the calibration wand, i.e. the diameter must be larger than the wand length.

When performing the calibration in an Extended calibration, it is very important to consider how the calibration wand is moved.

To achieve the highest accuracy, and indeed to make a successful calibration at all, it is vital that the calibration wand is moved considerably in the volume of the cameras that can see the reference object. It is even more important to move the wand in the volumes of overlap between different cameras. To ensure that you have time to do this increase the calibration time and also increase the **Maximum number of frames used as calibration input** setting on the **Calibration** page in the **Project options** dialog.

For example, if two cameras can see the reference object and three are positioned to extend the volume, it is very important to move the wand in the volumes of overlap between the two base volume cameras and the three extended volume cameras. The points collected in these volumes are the points used to calibrate the volumes relative to each other. If the wand is not moved enough in the volume of overlap, the extended volume of the three cameras will be very poorly calibrated relative to the base volume.

In addition to these volumes of higher importance, it is still important that all cameras can see the wand in as many orientations and positions as possible, just as in a normal calibration.

Frame calibration method

The Frame calibration method uses a structure (frame) with reflective markers with known and fixed positions. The frame is then placed in the measurement volume and a calibration is made on the stationary frame. The advantages with Frame calibration are that it is repeatable and invariable. The disadvantages are however that the resulting measurement volume is usually rather small and that the measurement error outside the volume is large.

Note: The frame calibration is only available for customers that bought QTM before 2.0.

Calibration frame

The calibration frame may be any rigid structure with known marker positions. The following conditions should be followed when constructing a frame:

- The frame must have at least five markers.
- The markers should be placed asymmetrically, since symmetry will make it difficult to calculate the correct position of the frame.
- The markers should not be in the same plane.
- The markers must be visible from many angles, since the cameras need to view the calibration markers simultaneously when calibrating.

A reference measurement must be made of the frame before it can be used so that the markers positions are known with a high accuracy.

How to use the calibration frame

Place the frame so that all of its markers are seen in the 2D views of every camera. Check in preview mode that every camera can see all markers. Then enter the settings on the **Calibration** page in the **Project options** dialog, see chapter “Frame calibration” on page 122.

It is also important to keep in mind that subsequent measurements should be made within the volume that is contained inside the calibration frame. This will ensure high accuracy in the measurement. However it is possible to capture markers outside of this volume, but it should be tested that the accuracy is acceptable.

When the calibration is finished, a measurement of the calibration frame can be performed to test the calibration. Compare the result of the measurement with the reference data of the calibration frame.

Fixed camera calibration method

The Fixed camera calibration method uses fixed locations of the cameras and the reference markers to calibrate a camera system. It can be used for systems covering large areas, where other methods are impractical. With this method a much larger measurement volume can be used, since one camera must not see the whole measurement volume. The disadvantage is however that the camera system must be fixed to its location.

Before the Fixed camera calibration is started the exact positions of the camera system and reference markers must be entered on the **Calibration** page in the **Project options** dialog, see chapter “Fixed camera calibration” on page 124.

Note: For more information about how to install and use a fixed camera system contact Qualisys AB about the QTM - Marine manual. They include detailed descriptions of the camera installation, survey measurement, fixed camera calibration, validation and the use of 6DOF bodies in marine applications.

Reprocess calibration

The calibration of a calibration file can be reprocessed to correct errors in the calibration. The following problems can be corrected:

- Calibration kit problems
- Coordinate system problems
- Incorrect definitions of calibration frames
- Incorrect positions of fixed cameras and reference markers
- Incorrect linearization files
- Incorrect **Calibration type**
- Deactivated cameras

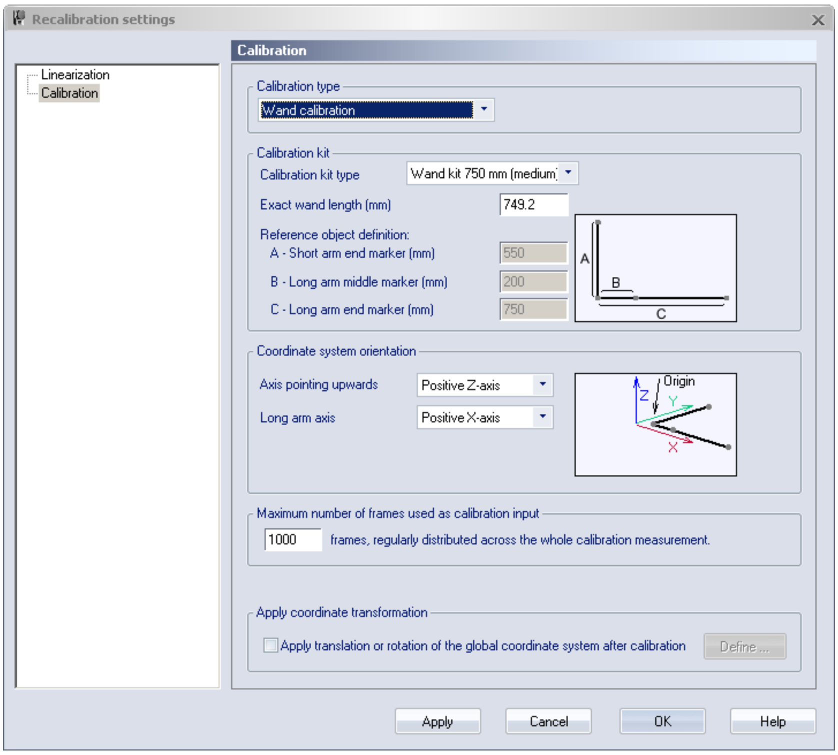
Note: If the camera system has not been moved it is better to make a new calibration and then reprocess the file with that calibration file.

Follow these steps to reprocess the calibration:

1. Open the calibration file of the wanted capture file. The calibration file is in the **Calibrations** folder in the project folder, see chapter “Project folder” on page 12.

Note: The name of the calibration file, of the current capture file, is found on the **Calibration** page in the **File reprocessing** dialog . To open the dialog click the **Reprocessing** icon , when the wanted capture file is opened.

2. Click the **Calibration** icon to open the **Recalibration settings** dialog.



3. In the dialog the **Calibration** and **Linearization** settings can be changed, see chapter “Calibration” on page 119 and chapter “Linearization” on page 116.

   Note: If a camera is deactivated on the **Linearization** page, that camera will not be used when a file is reprocessed with the calibration file. Activate the camera if you want it to be used in the measurements.

4. Click **OK** to start recalibration.

5. The **Calibration results** dialog is shown. Click **Use**if you want to use the reprocessed calibration as the current calibration. **OK** will only close the **Calibration results** dialog.

6. Save the calibration file and close the file.

7. All the capture files that have used this calibration must then be reprocessed, see chapter “Reprocessing a file” on page 281.

Capturing of measurements

Introduction to capture

Usually before you start a measurement it is best to open a new empty capture file with **New** on the **File** menu. The file will be opened in preview mode where you can check measurement volume and settings before starting the capture.

The capture is then started by clicking either the **Capture** icon or clicking **Capture** on the **Capture** menu. The settings needed to start the capture are set in the **Start capture** dialog, see chapter “Start capture” on the facing page. However if the system is calibrated and the settings are ok the new file is not necessary and you can click the **Capture** icon to open the **Start capture** dialog directly. The cameras will start in the same mode as they where in the last preview or measurement.

Note: When opening a new file QTM will detect automatically if the camera system has old firmware. The firmware must then be updated before the system can be used. For more information see chapter “Firmware update when opening a new file” on page 353.

The following capture related topics are also described in chapters below:

Batch capture
How to batch capture several files in a row.

6DOF real-time output
A description of how to output 6DOF data in real-time.

Measurement guidelines
Some guidelines on how to place the cameras and on the IR marker usage.

Outline of how to capture

The steps below are just an outline of how to perform a capture.

Follow these steps to do a capture:

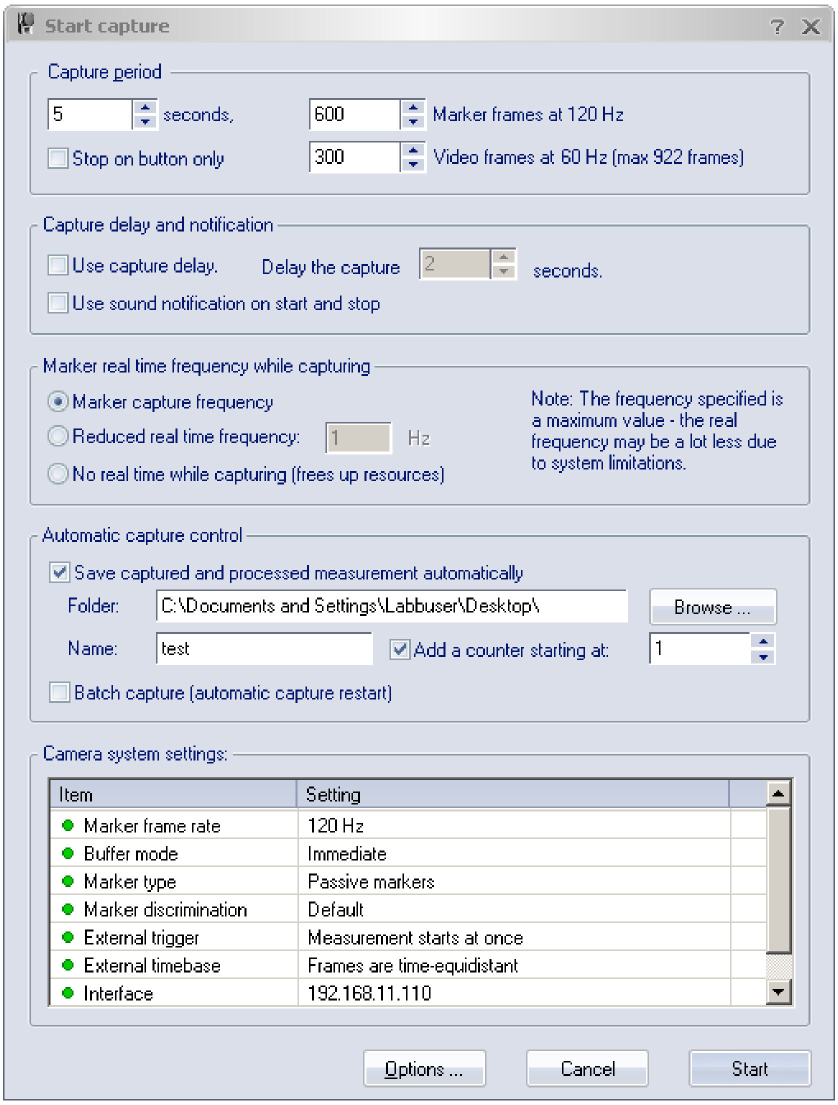
1. Switch on the camera system and start QTM.
2. Open a new file by clicking the **New file** icon. The cameras will start in the same mode as last preview or measurement.
   Note: If the system has been calibrated and is ready for a measurement you do not have to open a new file. Click the **Capture** icon and go directly to step 6.
3. Calibrate the system, see chapter “Outline of how to calibrate (Wand calibration)” on page 235.
4. Go the **Processing** page in the **Project options** dialog to activate any processing steps.
   Note: For information on how to define a 6DOF body see chapter “Outline of how to define 6DOF bodies” on page 288.
5. Click the **Capture** icon.
6. Specify the capture settings in **Start capture** dialog, especially the **Capture period**.
7. Check that all of the settings under the **Camera system settings** heading are correct.

8. Click **Start** to start the capture. When the capture is finished, the new motion capture file is displayed in QTM. The capture can always be stopped with **Stop capture** on the **Capture** menu or by clicking the **Stop capture** icon ■.

Note: If any problems occur during capture, check the settings in the **Project options** dialog and in the **Start capture** dialog. If that does not help, check the troubleshooting list in chapter "Troubleshooting capture" on page 390.

Start capture

The **Start capture** dialog appears before the start of every capture or batch capture.



Click **Start** to start a capture or click **Options** to change the settings in the **Project options** dialog.

Capture period

Under the **Capture period** heading, the capture period is specified in seconds or in number of frames. Fractions of a second can be specified, which will be rounded off to the nearest number of frames. If an Oqus camera is in Video mode the number of **Video frames**, the corresponding video capture rate and the maximum number of video frames are also displayed.

When measuring video the measurement time will be limited by the maximum number of video frames. The maximum number of video frames depends on the video image size, the image resolution and the internal memory of the camera.

With the **Stop on button only** option the capture does not stop until the **Stop capture** ■ button is pressed.

Capture delay and notification

Under the **Capture delay and notification** heading there are options for delaying the start of the capture (**Use capture delay**) and for sound notification on start and stop of the capture (**Use sound notification on start and stop**). When the **Use capture delay** option is used the delay is specified in seconds in the text box next to the option.

Marker real time frequency while capturing

While capturing a file QTM can still fetch and process the data and display it on the screen as well as making it available on the RT server. QTM will apply the same processing steps as under **Real time actions** on the **Processing** page in the **Project options** dialog. The following three options are available for frequency:

Marker capture frequency
The same frequency as the current **Marker capture frequency** on the **Camera system** page in the **Project options** dialog.
Note: The real time frequency can be lower than the **Marker capture frequency** for example because of too many markers at a too high frequency. If it is important with a stable real time frequency reduce the **Marker capture frequency** to a value where it works or use the **Reduced real time frequency** option.

Reduced real time frequency
Use a reduced real time frequency compared to the **Marker capture frequency**. If it is important with a stable real time frequency make sure that you test that the frequency is not too high for the current setup.

No real time while capturing
There is no real time processing done during the measurement, which means that nothing is updated on the screen.
Note: For ProReflex the fetching process can be faster with **No real time while capturing** if your graphics display is slow.

Note: If the Oqus system is in the frame buffering mode, the maximum real time frequency while capturing is 240 Hz. However you can of course still make measurements at higher frequencies.

Automatic capture control

Under the **Automatic capture control** heading there are options for automatic saving of measurement files and whether to use batch capture.

Select the **Save captured and processed measurement automatically** option to automatically save the measurement file. Enter the folder in which the files will be saved (**Folder**) and the name of the files (**Name**). An automatic counter can also be assigned to the filename.

When **Batch capture** is selected, QTM will make several consecutive measurements. When batch capturing **Save captured and processed measurement automatically** and the automatic counter must be selected. For information on batch capture see chapter “Batch capture” on the facing page.

Camera systems settings

Under the **Camera system settings** heading the measurement settings are displayed. The settings can be changed by right-clicking on the entry and then click **Change** or **Reset to default value**. The **Project options** dialog can also be reached by clicking **Options**.

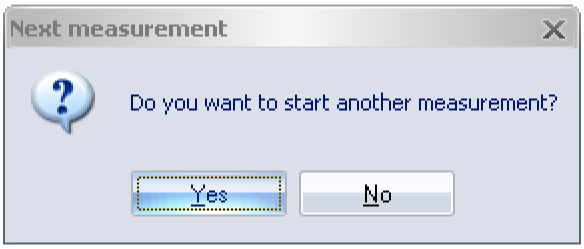
Batch capture

With batch capture, QTM will capture several consecutive measurements. Batch capture is activated with the **Batch capture** option on the **Start capture** dialog, before the start of the measurements. In this dialog, the options **Save captured and processed measurement automatically** and the automatic counter must also be selected so that the each measurement is saved in a separate file.

Before each new measurement in a batch capture QTM will wait for a start signal and the whole batch capture is stopped by a stop signal. These signals are given in different ways depending on whether external trigger is used or not.

External trigger
If the external trigger is used to start each measurement, to stop the batch capture you need to press Esc or click **Close** on the **File** menu. **Stop capture** on the **Capture** menu can be used during an individual capture to stop just that capture.

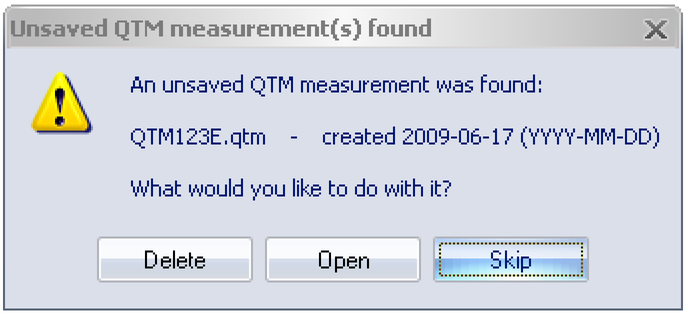
No external trigger
Start each measurement by clicking **Yes** in the **Next measurement** dialog. Stop the batch capture by clicking **No** in the dialog. **Stop capture** on the **Capture** menu can be used during an individual capture to stop just that capture.



Note: All of the processing steps that are selected on the **Processing** page in the **Project options** dialog will be performed before the next measurement can start.

Auto backup

The measurement can be saved automatically directly after the data is fetched with the **Auto backup** option on the **Processing** page in the **Project options** dialog. When activated a temporary file with the 2D data is saved before the other processing steps. If the file is very large the auto backup may take several minutes. Then if QTM crashes during the processing of the data the file can be retrieved at the next startup of QTM.



Measurement guidelines

The following chapters contains guidelines on how to best position the cameras, how to set the marker settings for the Oqus camera, how to use the camera flash groups and the usage of the IR markers.

Flash groups (ProReflex)

Camera positioning

Cameras must be mounted firmly on tripods or other stable structures, which isolate the camera from movements or vibrations of any sort.

2D motion capture

In 2D motion capture just a single camera is needed, which is positioned perpendicular to plane that is captured.

3D motion capture

To capture 3D data the camera system must consist of at least 2 cameras. The guidelines below can be used to set up a camera system.

- The best possible setup for a 3D motion capture system is to position it so that all cameras can see the L-shaped reference structure during the calibration, see chapter “Calibration of the camera system” on page 235.
  Note: The cameras can be positioned so that just two of the cameras are able to see the calibration reference object. The rest of the cameras must then overlap each other’s field of view (FOV) to be able to calibrate the system. For this setup QTM will automatically use the Extended calibration method, see chapter “Extended calibration” on page 242.
- To reconstruct 3D, data at least two cameras must see each marker during the measurement. Therefore it is best to position the cameras so that as many cameras as possible see each marker during the measurement.
- The angle of incidence between any two cameras should ideally be more than 60 degrees and at least more than 30 degrees. The accuracy of the 3D data calculated from only two cameras placed at less than 30 degrees can degrade below usable levels.
- In order to avoid unwanted reflections, position the cameras so that every camera’s view of flashes from other cameras is minimized. E.g. put the cameras above the measurement volume so that the cameras have an angle of about 20 degrees in relation to the floor.
- Obviously the cameras must also be positioned so that they view the volume where the motion of the measurement subject will occur. Mark the volume by putting markers in the corners of the base of the measurement volume. Then make sure that all cameras can see all of the markers by looking at a **2D view** window in preview mode. Preferably, the cameras should not see much more than the measurement volume.
  Note: Remove the markers in the corners before the actual motion capture is started.

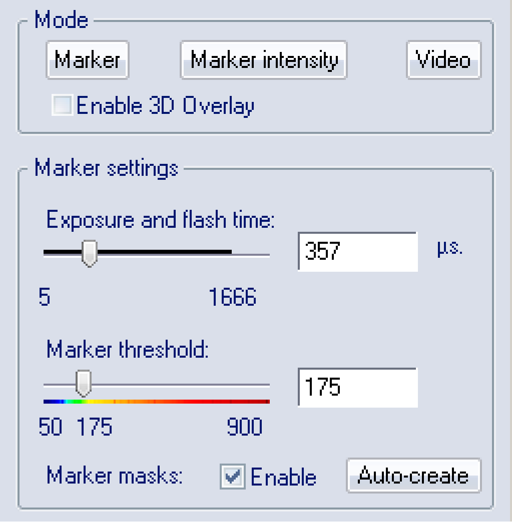
Tips on marker settings in QTM

The two important settings for marker calculation are **Exposure time** and **Marker threshold** . Usually it is sufficient to change them to see the markers in QTM. However it is not possible to give exact advice on how to set the settings, because the relations are too complex. The tips below will show you the basic method of how to use the settings. There are also other settings that can help to get a better measurement, e.g. **Marker masking** and **Marker discrimination**.

First of all make sure that the focus and aperture are correct, see “Setting the aperture and focus” on page 343.

Exposure and threshold must be used together, because changing one can lead to that you have to change the other. Follow these steps to set exposure and threshold settings.

1. Start a new measurement and stay in RT/preview mode. Use the **Camera settings** sidebar to change the settings.



2. Set the **Exposure time** to 400 microseconds (200 µs for Oqus 3+ and Oqus 4) and **Marker Threshold** to 175. These are the default values and often it is a good starting point.
3. Start with looking at the marker intensity image. Click on the **Marker intensity** button in the **Camera settings** sidebar to show the marker intensity image. For more information see “Video preview in QTM” on page 256.
4. If the markers are not bright red in the marker intensity image, try increasing the **Exposure time** until they are bright red. For example with a longer distance to the markers, you may need a longer exposure time.
    a. For higher frequencies it might not be possible to increase the exposure so that the markers are bright red. However, as long as they are brighter than the background the camera should be able to find them by lowering the marker threshold.
    b. If there are extra reflections, you can try reducing the **Exposure time**. Extra reflections are anything that is not a marker and has a color different from blue. Green is the threshold level.
    c. Remember that you can set this setting individually for each camera, see “Camera settings sidebar” on page 40.
5. Switch back to Marker mode and check if the markers are visible. If they are not, go back to Marker intensity mode and adjust the **Marker threshold** value. It is not possible to give an exact value for the threshold at a certain exposure, because each setup is different.

   a. Increase the threshold if the background looks light blue or even green in the marker intensity image. A light blue background will make it harder for the camera to calculate the markers.
   b. Decrease the threshold if the markers are not bright red in the marker intensity image. For example, at short exposure times of 200 µs (100 µs för Oqus 3+ and Oqus 4) and lower the threshold needs to be low, usually around 100-150.
   c. Make sure that the marker calculation is stable at the selected threshold. If the threshold gets too low there will be a lot of extra reflections or no markers at all. A too high threshold will result in small markers or missing markers.
   d. Remember that you can set this setting individually for each camera, see “Camera settings sidebar” on page 40.
6. Finally check that the markers are large enough to give a good accuracy. Check that the marker size is at least 200 in the **Data info** window.

Delayed exposure to reduce reflections from other cameras

Exposure delay can be used in environments where the strobe light from one set of cameras disturbs marker capture in other cameras, and all other methods of discriminating the markers have failed. Exposure delay will cause the time of exposure to be shifted among different cameras. For example, a 6-camera system has three cameras located on one side and three on the other side of a room, with a highly reflective floor, where the strobe from cameras on one side disturbs cameras on the other side. Assigning the first three cameras to one exposure group and the other three cameras to another exposure group will eliminate the problem of strobe interference. The downside is that the shift in exposure time can cause tracking problems, especially on fast moving markers. Therefore it is important to first consider the following changes to the setup.

1. Test a different camera positioning. The reflections can be reduced if the cameras do not look straight at each other.
2. Can anything be done to the floor or any other reflective area of the room to make it less reflective.
3. It can also help to reduce the exposure time or increase the marker threshold, because the unwanted reflections are usually less intense than the markers.

Follow these instructions to use the delayed exposure.

4. To use the delayed exposure you must first find out which cameras are causing reflections in other cameras. It is usually best to use the marker intensity mode to see where the reflection comes from.
5. When you know the cause of the reflections you place the cameras in different groups on the **Advanced camera settings** page in the **Project options** dialog, see chapter “Exposure delay” on page 150.
   a. Activate the **Exposure delay mode** called **Camera groups**, which will calculate the delay for each group automatically by comparing it to the longest exposure time of the cameras in the previous group. Do not use the **Advanced** mode unless you are absolutely sure of what you are doing.
   b. Select the cameras that you want in the group from the list to the right.

c. Then select the group on the **Camera groups** setting. Make sure that you always start with group 1 and continue with 2 and so on.

d. Repeat steps b and c for each group that you want to create. Usually for a setup with cameras on two sides of the volume it is enough with two groups, one for each side.

e. When the exposure delay is activated **Using delayed exposure** is displayed in the status bar.

6. To check which groups the cameras are in, go to the 2D view and check the delayed exposure setting next to the camera number.

Because the exposure time is shifted the markers will have moved slightly between frames of the different camera groups. Therefore the true 2D positions of the markers in the delayed cameras are calculated by predicting their position. This keeps the data quality from degrading for long delays and fast movements. However the first 3 frames of each camera cannot be predicted, so if the markers are often hidden in the cameras there can be a lot of ghost markers.

IMPORTANT: Since the compensation for the 2D position is not used in the calibration, you can get much higher residuals if exposure delay is turned on. QTM will warn you of this before the calibration and you can select whether to use it or not in the calibration. However you might still have to mask out the extra reflections if it is necessary. If the delayed exposure is turned on you have to move the wand very slowly during the calibration.

Note: Delayed exposure only works for Oqus 1-, 3- and 3+-series. This is because the 5-series camera has a rolling shutter, which means that some part of the image is always being exposed.

Note: The time of exposure will return to the default value, if the camera is in video mode.

Note: Reflections that come from the flash of the camera itself can of course not be removed with a delay. Then you have to either cover the reflective material or try reducing the exposure time.

Flash groups (ProReflex)

There are three flash groups in the ProReflex MCU: outer, middle and inner group. Activating the different flash groups on the **Camera settings** page in the **Project options** dialog will adjust the flash intensity and the strobe volume, so that the camera is optimized for different distances. The following selections of flash groups are recommended with respect to the distance between cameras and markers:

Distance	Flash group
0 – 2.5 m	Inner
0 – 4 m	Inner + Middle
0 – 30 m	Inner + Middle + Outer

Note: Except for adjusting the camera system to different distances, flash groups can also be used to compensate for differences in the intensity between the markers attached to the calibration object and the markers used in the measurement.

IR markers

To make a measurement some kind of IR-reflective or IR-transmitting material is needed. In the sections below there are some guidelines on how to use and apply the markers.

Valid marker types

There are two marker types: passive and active markers. These can be found in several different sizes. Which marker type and size to use depends on the purpose and the measurement conditions, for further guidelines on how to choose the size see chapter "Marker size" below.

Passive/Active markers

The difference between passive and active markers is that the passive marker reflects IR-light, while the active marker transmits IR-light.

There are two types of active markers: long range active markers and Close range active markers. The types can be used either in synchronized or continuous mode. Synchronized markers are controlled by the IR flash of the cameras. Continuous markers on the other hand does not need any IR flash at all.

Note: Synchronized active markers cannot be mixed with passive markers.

Standard marker sizes

The sizes of standard passive lightweight markers are:

Size	Shape
5 mm	Hemispherical
4 mm	Hemispherical
7 mm	Spherical
12 mm	Spherical
19 mm	Spherical
30 mm	Spherical
40 mm	Spherical

The standard active marker is:

40 and 50 mm, spherical with internal or external power supply

There are also other sizes of markers available. Contact your distributor or Qualisys AB for further information.

Marker size

It is important to choose the correct marker size, since a larger marker gives a larger area to use to determine the central point of the marker. The central point data is the input to the tracking of the markers movement and is therefore very important.

Rules to determine which marker size to use are:

- Use as big markers as possible without the markers being in the way or restraining normal movement.
- If the markers are attached very closely to each other, use as large markers as possible without the markers merging in the 2D view of the cameras. If the markers merge, the central point calculation of these markers will be ambiguous.
- To achieve a good accuracy the size of the markers in the 2D view should be at least 200 subpixels for Oqus cameras. For ProReflex the recommended size is 7 camera units (per mill of the diagonal FOV). Check the size of the markers in the 2D view for each camera in the**Data info** window. Right-click

in the **Data info** window and click **Display 2D data** to choose the camera. If the size is smaller the central point calculation will be poor.

Marker placement

The markers should be placed in a way so that they are visible during as much of the measurement time as possible. Check that clothes or rotating objects, such as body parts, do not move in a way that hides the markers from the camera.

Marker maintenance

The passive markers should be attached to the object by double adhesive tape.

Warning: Do not use glue or other types of chemical substances to attach the markers to the object as it may damage the markers.

The passive markers can be cleaned from grease and dirt with soap and water. Do not use very hot water when cleaning the markers and do not use a brush when cleaning the markers.

Note: Be careful when cleaning the markers, otherwise the reflective material on the markers can be damaged. If part of the reflective material is damaged on the passive marker, it should be discarded because the central point calculation of this marker will be indeterminable.

Oqus video capture

Introduction to high-speed video

An important feature of the Oqus camera is the high-speed video functionality. This means that you can record regular video frames with the Oqus camera. It is also possible to switch to Video mode in preview to watch the field of view of any Oqus camera. Oqus 3-, 3+- and 5-series with high-speed video capabilities can capture video up to 10000 fps, while Oqus 1-series can capture up to 1000 fps. The high-speed cameras have clear glass and the second number in the Oqus type is 1, e.g. 310. The standard cameras is limited to 30 Hz, the standard version has a black glass and the second number in the Oqus type is 0, e.g. 300.

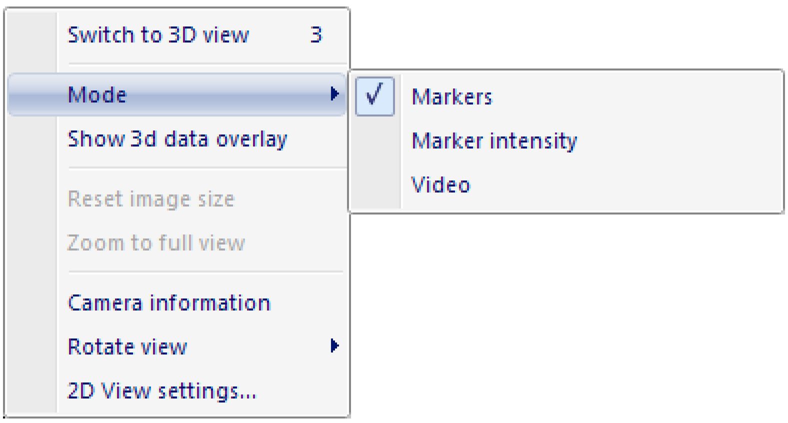
The Oqus camera captures video frames that are synchronized with the marker frames in other Oqus cameras in the same system, however it is possible to use different capture rate for marker and video. If the Oqus camera has the high-speed functionality it is possible to capture video within the same limitations as the marker capture, so the video and marker capture rates can be the same. For a standard Oqus the video capture is limited to 30 Hz and the dark glass filters out the visible light, therefore the video functionality in the standard Oqus is mostly used in preview.

The Oqus video settings are controlled on the **Advanced** page in **Project options** dialog, see chapter “Advanced (Oqus)” on page 143.

Video preview in QTM

The video preview is a helpful functionality in the setup of the camera system. It can for example be used to find reflections or look at the current field of view of the camera. Any Oqus camera can be changed to show video image in preview. Follow these instructions to get the best image in preview. For the instructions for making a high-speed video see chapter “Outline of how to capture high-speed video” on page 258.

1. Open a new measurement and open a 2D view window.
2. Click on **Video** or **Marker intensity** in the **2D View window** menu to switch the camera to Video mode or use the **Mode** options in the **Camera settings** sidebar to switch all visible cameras to **Video** or **Marker intensity** mode. When switching to video preview the RT frequency will automatically change to variable because of the increased amount of data. The RT frequency will be a few Hz and depends among other things on the number of cameras.



3. There are two different Video modes.

   Marker intensity
   In the Marker intensity mode the cameras use the same Exposure and flash time as for markers. The video is also color coded, see image below, so that it is easier to see the **Marker threshold**. The **Marker threshold** will always be where the color is green, therefore the image will change when you change the **Exposure time** or the **Marker threshold**. Which means that anything yellow and red in the image will be markers, when you switch back to Marker mode.

   Video
   In the Video mode the cameras use the exposure and flash time from the **Video settings** on the **Camera settings** page. This is the mode that you use when you want to capture video with the Oqus camera.

4. To optimize the preview frequency the number of pixels in the image are automatically changed depending on the 2D view size and the zoom.

5. Use the Video mode and set the following settings on the **Camera settings** page to get a bright image in preview with non high-speed Oqus cameras.

   Capture rate
   Set the **Capture rate** to 10 Hz.

   Exposure time
   Set the **Exposure time** to 99000 microseconds for a standard camera. If the image is still too dark lower the **Capture rate** and increase the **Exposure time** even more.

   Flash time
   The flash does not change the brightness of the image a lot on long distances. Therefore the **Flash time** can be set to 0. If you want to see the markers better, e.g. when setting the focus on a marker, you can set the flash time to about 400 microseconds.

6. Then to see more of the video image double-click on the camera so that only that camera is shown in the 2D view window. It is also possible to zoom in the image, see chapter “Modifying the 2D view” on page 39.

7. The following list are some of the things that the Video preview can be used for.

   Check the cause of extra reflections
   Zoom in on the extra reflection in Marker intensity mode if necessary.

   Check the camera field of view

   Check the focus setting
   Use the Video mode with full resolution to focus.

High-speed video capture

A Oqus camera with High-speed functionality can capture high-speed video from QTM. This means that it is possible to capture video from Oqus, which is synchronized to the motion capture data. The video data can then be viewed in QTM and exported to an AVI-file, which can be processed in video processing software.

The video frame rate has the same limitations as the marker capture according to the following table.

	Max frame rate full resolution (Hz)	Max frame rate reduced resolution (Hz)
Oqus 1-series	250	1000
Oqus 3-series	500	10000
Oqus 3+-series (high speed mode)	500 (1764)	10000 (10000)
Oqus 5-series	180	10000

There are two main issues that must be considered when capturing high-speed video.

Lighting
The first main issue with the video capture is the lighting. To capture video data the IR filter has been removed from the Oqus camera, i.e. a clear glass has been mounted. Up to about 60 Hz it is often possible to capture video without extra lighting, however it depends a lot on how light the room is.
For high speed video, meaning frequencies above 100 Hz, extra lighting is definitely needed. High intensity lamps must be used and with increasing frequency and distance you need more light intensity.
Note: If you have mounted the external IR filter on the lens that must be removed, so that the visible light is recorded.

Data storage
The other issue is the large amount of data that will be captured by an Oqus camera in video mode. The data must be stored in the camera during capture, this means that the maximum amount of data that can be captured is 1.1 GByte and that it takes time to download the video from the camera. The amount of data that can be captured depends on the frame rate and the image size, the table below shows the capacity at full resolution. For another resolution multiply the amount of data with the reduced resolution/full resolution.

	Maximum video buffer capacity (full resolution and frame rate)	Maximum video buffer capacity (full resolution)
Oqus 1-series	15.2 s	3800 frames
Oqus 3-series	1.8 s	900 frames
Oqus 3+-series (high speed mode)	1.8 s (2 s)	900 frames (3600 frames)
Oqus 5-series	1.4 s	290 frames

Outline of how to capture high-speed video

The following outline describes how to capture high-speed video with an Oqus camera.

1. Switch on the camera system and start QTM.
2. Open a new file by clicking the **New file** icon. If you want to use the 3D overlay functionality it is important to check that the camera system has been calibrated.

3. Switch to 2D view if it is not visible, i.e. right-click in the View window and select **Switch to 2D view** from the menu.
4. Right-click on the 2D view for the camera that will capture high-speed video. Select **Mode/Video** to switch to Video mode. The 2D view for that camera will switch to a video image.
5. Open the aperture and set the focus.
6. Change the settings for Video capture in the **Camera settings** sidebar.
    a. Set the video **Capture rate**. This can be set independently of the marker image rate.
    Note: The image size is reduced automatically when it is changed on the **Camera settings** sidebar.
    b. Set the **Exposure time** to a value that makes the image bright enough, test until you are satisfied.

    If you have no extra light the exposure time needs to be quite high, at least 16000 microseconds or even up to 40000. This limits the capture rate that can be used. It also means that fast movements will be blurred.

    For high capture rates and measurements with fast movement, extra lighting is needed because the exposure time must be extremely short, sometimes as short as 100 microseconds.

    Use a Codec if you want to reduce the size of the avi-files. If the file will be used in analysis it must not be reduced too much as it can influence the analysis.
    Note: The codec does not change the download speed.
    Note: The **Image format** will reduce the image directly in the camera, but then you will lose pixels.
7. A video capture takes about a second to initialize so it is recommended that you use external trigger and pretrigger to start the capture. For the settings see chapter “Timing” on page 130.
8. Click the **Capture** icon and set the capture time and other capture settings. Keep the capture time as short as possible to reduce the fetching time.
9. Click OK and then press the trigger button when you want to start the measurement.
10. Wait for the fetching of video data to finish. Depending on how large file you have captured and the number of cameras, it can take up to some minutes to fetch the data.
11. The video file is displayed in the 2D view window and can be played in synchronization with other data.
    Note: If the video data is uncompressed it is not possible to play the file in normal speed as the video playback will be too slow. Then it is better to step through the file instead.

Oqus video files

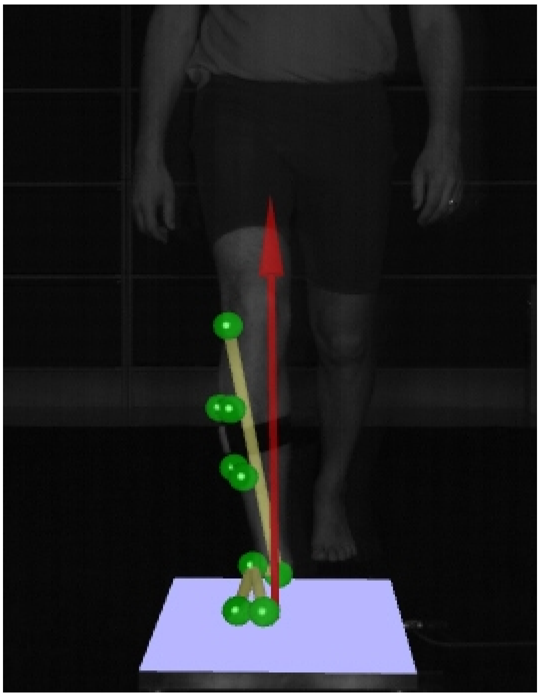
Video files captured by an Oqus camera are saved as an AVI file. The file will be saved in the same location as the QTM file and named the same but with the serial number of the camera at the end.

The video file i displayed in the 2D view window in QTM, but it can also be opened in any video software, for example Windows Media Player.

Note: If the file is uncompressed it will not play at the correct speed in any program. This is because the amount of data is too large for graphic board to handle.

Note: If you have compressed the video file with a codec the computer where you play the file must support have that codec installed. For information on video codec see chapter “Codecs for Oqus high-speed video files” on the facing page.

3D data overlay on video



The 3D data can be overlayed on the Oqus video data to show the 3D view from that camera's viewpoint. This can for example be used for showing the force arrow in the video of someone stepping on a force plate. Which 3D objects that are displayed in the 3D overlay is optional.

The Oqus video will always be synchronized with the marker data. However if the video capture rate is different from the marker capture rate the video will of course only be exactly at the same position for corresponding frames. For example with video capture at 30 Hz and marker capture at 120 Hz, the video data will be updated every fourth marker frame.

Follow these steps to activate the 3D data overlay.

1. Calibrate the camera system, including the Oqus cameras that are used in video mode.
2. Open a **2D view** window in RT/preview mode or in a file.
3. Right-click on the camera where you want to turn on the overlay and select **Show 3D data overlay**.
4. The 3D elements displayed in the overlay and the opacity can be changed on the **2D view settings** page in the **Project options** dialog, see chapter “2D view settings” on page 220.
5. The video data display in the 2D view can be switched between linearized and unlinearized on the **2D view settings** page. To match the video with the 3D data the data must be linearized.

Codecs for Oqus high-speed video files

A codec can be used to dramatically reduce the file size of the high-speed video files. Any codec that can run on Microsoft DirectShow can be used in QTM to compress Oqus high-speed video files. However as there are a lot of codec we have not tested them all and the three below are the ones we recommend. All of them must be downloaded from the internet. They are free but we are not allowed to bundle them with QTM.

Lossless codecs keeps all original image information in the files which makes them suitable for high quality video analysis. Lossy codec removes information, usually based on what the human eye can register, which makes a video compressed with a lossy codec suitable for movie encoding.

The codec of the high-speed video file must be set before the measurement is started, either by right-clicking on an Oqus camera in Video mode or with the **Video compression** settings on the **Advanced** page in the **Project options** dialog. It is not possible to compress a saved video file in QTM. However QTM will still be able to play the file if it is compressed in another program.

Important: Even for the codecs we recommend we will not guarantee that the files can be played in any program, even if the codec is installed on the computer. This is because every program is a little bit different in how they play the file and it is impossible for us to test all of the settings and configuration. Therefore we recommend that you make tests with the codec before making the measurements.

Lagarith

- A lossless codec with a typical compression of 4-8 times. The codec uses arithmetic and RLE encoding.
- In QTM the files can be played close to the original speed.
- Lagarith works well for analysis with Tema QVA, however the playback speed is slower than for example Indeo but the data is on the other hand unchanged.
- The codec can be downloaded from http://lags.leetcode.net/codec.html.

FFDS (MJPEG)

- FFDS is a codec package with a lot of different codecs. The codec we recommend is the MJPEG codec. The other codecs have not been tested with QTM.
- A fast, lossy codec that at max quality has a compression of about 15-30 times. The codec uses JPEG compression of each video frame. It is important to use the correct codec settings see chapter “FFDS-MJPEG codec settings” on page 154.
- In QTM these video files can be played at the original speed.
- The codec works well with Tema QVA, both playback speed and quality is ok for a lossy codec.
- The codec can be downloaded from http://www.free-codecs.com/download/ffdshow.htm.

Analyzing data in video files

Special video analysis software must be used to analyze data in a video file. The only requirement is that it can open the AVI file.

Oqus high-speed cameras are delivered with a 60-days demo of TEMA QVA (Qualisys Video Analysis), see chapter “TEMA QVA” below. With TEMA QVA the data can be tracked and analyzed, for information on how to use the different functions in TEMA QVA refer to the TEMA manual included in the installation.

A special function for QTM with TEMA QVA is that the tracked video data can be exported back to QTM and tracked in 3D. Follow these instructions to 3D track the video data.

1. Calibrate the Oqus camera system with all cameras in marker mode. In this way the camera positions will be known in the capture files.
2. Capture a file where some or all of the cameras capture video.
3. Open TEMA QVA and open a new camera view.
4. Locate the file for a camera and open it.
5. Check that the frame rate is correct.
6. Track the points that you want in the file. With TEMA QVA a new file must be started for more than 5 points.
   Note: Do not change the coordinate system.
7. When the tracking is finished, open a new **Time table** on the **Diagram** menu.
8. Right-click on the grey part of the empty table to open the **Time Table Properties** dialog.
   a. Add the position of the points by double-clicking on them. Then click **OK**.
9. Right-click on the list and then select **Export** on the menu. Save the file as a TXT file, include the camera serial number in the name as that will help when importing the data in QTM.
10. Repeat steps 4-9 until all points that you want have been tracked.
11. Open the QTM file that was saved with the video files.
12. Import the data with **File/Import/Tema 2D file**. Open a file and then select the correct camera for that file.
    Note: You can import data several times to the same Oqus camera, meaning that you can track more markers than 5 if you track them in separate files in QVA. Right click on a Video view of an Oqus camera and select **Delete 2D data** to delete all of the previously imported QVA data.
13. Repeat steps 11-12 until all files have been imported.
14. The file can now be tracked and processed just like regular QTM file.

TEMA QVA

With every shipment of a Qualisys High Speed Video Camera (Oqus), a 60-day trial version of TEMA QVA will now be included for the user to install and use at no extra charge.

TEMA QVA (Qualisys Video Analysis) is an analysis software tool used to manage and report video data. Together with a Qualisys Motion Capture High Speed Video System, QVA provides an advanced and affordable solution for biomechanical motion analysis. The high speed video image can now be evaluated both visually, by watching the sequence in slow-motion, as well as analytically, by means of QVA. TEMA QVA is developed around and based upon TEMA, software created by Image Systems, one of

Qualisys global strategic partners in developing and marketing world-class motion analysis systems.

TEMA QVA can track a number of points throughout the image sequence and the results can be presented in a variety of predefined graphs and tables. Depending on the requirements of the user, QVA is offered in different versions, each with its own unique features. The standard version contains following features.

Tracking	Correlation
Number of points in one session	5
Import of image	AVI, TIFF, MPEG, JPEG and others
Export of diagrams and images	To Word document
Scaling	Dynamic, static and manually scaling
Coordinate system	Visualizing of grid system/division and scale of image
Diagram	X/Y, X/T diagram, full interactivity
Tables	Free choice of parameters, full interactivity
Toolbox	Several options except printing parameters, scales of diagrams printing of logo types, test comments etc.

Active filtering for capturing outdoors

How to use active filtering

The active filtering mode is most useful in daylight conditions, because it filters out background light. This is done by capturing extra images without the IR flash these images are then used to remove the background light from the images used by the camera. Active filtering is available on 4-series, 3+-series and 1-series cameras.

The standard and recommended setting is the **Continuous** mode in which an extra image without IR flash is capture before each actual image. Therefore the maximum frequency is reduced by 50% compared to the normal maximum frequency. It is also not possible to increase the capture rate by reducing the image size, for example on the 1-series the maximum frame rate is 115 Hz.

Note: Active filtering does not help if you have problem with extra reflections from the IR strobe of the cameras. In that case you must try to remove the reflective material or use the delayed exposure setting, see chapter "Delayed exposure to reduce reflections from other cameras" on page 252.

Follow these instructions to use active filtering.

1. Set up the camera system and focus the cameras.
2. Turn off the **Active filtering** option in the **Camera settings** sidebar and look at the marker intensity mode.
    a. Use a quite low **Exposure time** around 200 µs. Then change the exposure time so that the markers are just at the maximum end of the red color.
3. Turn on the **Active filtering** option and check that the markers looks OK in preview. It is best to do this check on the subject that you are going to measure and not on static markers.
    a. If it does not look OK, try to increase the **Marker threshold** setting until it is better.
4. Continue to calibrate the system as usual.

Oqus marker masking

How to use marker masking

The marker masking is a tool in Oqus to delete unwanted markers in the 2D data. The area is defined per camera and is best defined manually in the 2D view window. For an explanation of the auto marker masking functionality see chapter "How to use auto marker masking" on the facing page. The masks can also be defined manually on the **Advanced** page in the **Project options** dialog. For explanation of the settings see chapter "Marker masking" on page 146.

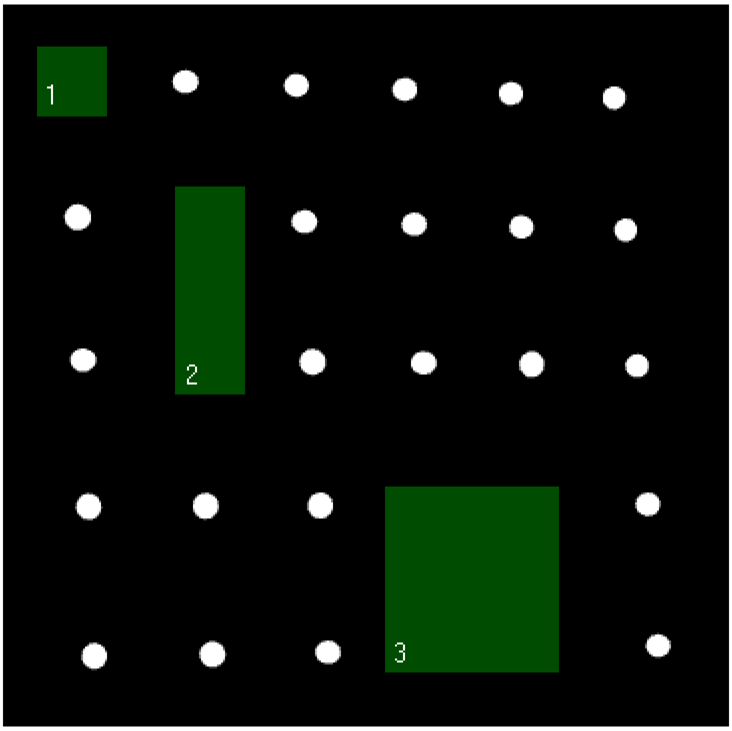
Follow these steps to add a masking area to a camera.

1. Open a new file by clicking the **New file** icon.
2. Open a **2D view** window so that you can see the 2D markers in preview.
3. View the cameras in either Marker or Marker intensity mode.

4. Select the **Masker mask** tool in the **2D view** toolbar and draw a mask over the area with the unwanted marker.

   The mask can be resized and moved by placing the cursor and hold down the mouse button on the edges of the mask respectively on the mask. Delete the mask by right-clicking on the mask and select **Delete this mask**.

   Repeat this step until all of the unwanted markers are covered. There can be up to 5 masks per camera.



How to use auto marker masking

The auto marker masking can be used to remove unwanted static reflections automatically. Follow this procedure to apply the auto marker masking.

1. Set up the camera system so that it covers the correct volume.
2. Check if you have any unwanted markers in the cameras. Before using auto marker masking try to remove the physical cause of the marker. Also try changing the **Exposure** and **Marker threshold** settings to remove the markers.
3. If the unwanted markers cannot be removed, make sure that you are in preview mode and then open a **2D view** window.
4. Select the cameras where you want to add marker masking with the camera buttons. You can select all of the cameras or just some of the cameras.
5. Click the **Auto-create** button on the **Camera settings** sidebar to open the **Create marker masks** dialog.

Click **Start** to get the position of the unwanted markers.

Note: There can be 5 marker masks per camera, if there are more than 5 in one camera the bottom unwanted markers will not be covered by masks.

Note: If there already are masks on the camera you will have to choose whether to remove the old masks or not. Answer **Yes** if you want to remove them and add the new masks. Answer **No** if you want to keep the old masks and append the new ones to the list.

6. To see that the masks really cover a marker you can deselect the **Enable marker masks** checkbox. The masks will then be inactived and the markers below will appear.

Oqus marker filtering

How to use circularity filtering and correction of merged markers

The circularity filtering and correction of merged markers can be used to detect bad marker data and to correct for markers that are considered to be merged. The quality of 3D data and how to use the filter depends a lot on the number of cameras in the system. If you have three or more cameras covering the same markers, it is usually better to try to remove the non-circular markers. Because then you have enough data to create the 3D data anyway. However if there is usually just two cameras viewing the same markers the data can become more complete by correcting for merged markers. This will result in more 3D data, which otherwise cannot be calculated.

The settings for the Qqus camera is found on the **Advanced (Oqus)** page in the **Project options** dialog, see chapter “Marker circularity filtering and correction of merged markers” on page 148.

The markers used by the filter are filtered out by the camera depending on the **Circularity level** setting. 100 means that all markers are accepted as circular by the camera, while 0 means that all markers are marked as non-circular by the camera. For advice on how to set the level see the instructions for the different modes below.

When you turn on the filtering the markers will be color coded in the 2D view window according to his list:

> **White - Circular markers**
> These markers are below the **Circularity level** and are therefore not handled by the filter.
>
> **Light Yellow - non-circular markers used by the 3D tracker**
> These markers are detected as non-circular by the camera, but are not considered to be merged. It can only appear with the **Correct merged markers** option.
>
> **Dark Yellow - non-circular markers that are not used by the 3D tracker**
> These markers are not used by the 3D tracker. It can only appear if you select **Correct merged markers, remove others** or **Remove all non-circular markers** options. These markers are not shown in the **Data info** window.
>
> **Green - corrected markers**
> These markers have been detected as merged and then corrected by the filter.

How to use the marker filtering and correction depend a lot on the camera setup. One thing that must be considered is the 2D size of the markers. To be used by the filter the marker has to be larger than 320 in 2D y-size. Therefore all markers below this size are considered as circular markers. Use the following advice to set the settings for the different modes.

> **Correct merged markers**
> Any markers that have been filtered out by the camera as non-circular will be checked if they are merged by the filter. Merged markers are corrected and the other non-circular markers are kept for the 3D tracker.
>
> > **Circularity level**
> > In this mode you can keep the **Circularity level** quite high. The default level is 50 and that usually works.

Note: Changing the Circularity level will not change how QTM detects merged markers. It will only change which markers will be checked by the filter for merged markers.

Effect on 3D data
This mode will give you more 3D data than the unfiltered data, because the corrected merged markers can be used to calculate the 3D markers. It has most effect if only two cameras can see the marker. The data may however be less accurate if the marker size is close to 320, because then it is hard to calculate the actual marker center.

Note: Corrected merged markers are always used by the 3D tracker. Uncorrected non-circular markers will not be used by the 3D tracker if two other cameras with circular markers can see the marker.

Correct merged markers, remove others.
Any markers that have been filtered out by the camera as non-circular will be checked if they are merged by the filter. Merged markers are corrected and the other non-circular markers are not used by the 3D tracker.

Circularity level
In this mode you can keep the **Circularity level** quite high. The default level is 50 and that usually works well to find markers that are merged. However if you want to filter out more or less markers you can decrease respectively increase the level. Check the 2D data for dark yellow markers to find out which markers are not used by the 3D tracker.

Effect on 3D data
This mode will probably give you slightly less 3D data than the unfiltered data, because the uncorrected non-circular data is not used by the 3D tracker. Compared with the **Correct merged markers** mode there will be less 3D data, but it will also be a more accurate in some cases.

Remove all non-circular markers
All markers that have been detected as non-circular by the camera are removed from the 3D tracker.

Circularity level
Check the dark yellow markers in the 2D data to find out which markers are not used by the 3D tracker. Then set the level according to how much you want to delete.

Effect on 3D data
This mode will give you less 2D data than the unfiltered data. However if you have many cameras that can see each marker you will not loose much 3D data and it will also be more exact.

Real time mode

Real time in QTM

Introduction to real time

The QTM real time process enables QTM to send data to any program that can receive data on TCP/IP or UDP/IP. The data can either be accessed through Qualisys own RT protocol or with OSC.

At the moment there are interfaces for iOS devices, Visual3D, Motion Monitor, Motionbuilder, LabView and Matlab. Please contact Qualisys AB if you want to use these interfaces. If you want to transfer real time data into other programs the protocol and examples are included in the QTM installer. The files are locates in RT protocol folder in the Qualisys Track Manager folder. If you cannot find the files run the QTM installer again and make sure to select the real-time protocol and examples. If you have any questions please contact Qualisys AB.

The real time performance depends on the computer, the number of cameras and the number of markers. To get the best performance, use a dual-core desktop computer with at least 2 gigabyte RAM and a good graphics card.

How real time works in QTM

QTM will process the 2D in real time as soon as a measurement file is open in preview, i.e. when you click on **New** in the **File** menu. The following processing steps can be done in real-time, depending on the settings on the **Processing** page in the **Project options** dialog.

3D tracking

Apply the current AIM models

Calculate 6DOF

Calculate force data

This data can then be viewed in QTM and can also be sent to another computer via TCP/IP. The TCP/IP server is always active and waits for a connection from another program. The following data can be sent via TCP/IP. For more detailed information check the RT protocol file in **RT Protocol** folder that is installed with QTM.

3D data (Identified, Unidentified, Residuals)

2D data (unlinearized, linearized)

Analog data (Analog boards and EMG)

Force data (Force, Moment and COP)

6DOF data (Position, Rotation matrix, Euler angles, Residuals)

Parameters (e.g. Label name, Statistics for the RT performance, Force plate settings)

Events

Images (from cameras in Marker intensity and Video mode)

QTM can also be controlled via the RT protocol, e.g. to start/stop measurements and to change camera mode.

OSC data, contact Qualisys AB for more information.

These are the different steps in the real-time process.

1. The cameras capture data at a fixed frequency.
2. RT processing of data in QTM which is performed as fast as possible.
3. Data is sent on the TCP/IP output. If QTM takes too long time processing a frame, the next frame captured by the cameras will not be processed at all. This is shown in the main status bar as a decreased RT frequency.

The real-time marker frequency is set on the **Camera system** page in the **Project option**s dialog. The analog and EMG sample rates depend on their respective setting. The analog and EMG is started in synchronization with the markers. The analog boards are always started in synchronization with the sync out signal. The EMG on the other hand needs to be started with the trigger button to be synchronized, see chapters ”Noraxon QTM settings” and ”Mega QTM settings”.

All of the analog, EMG and force samples are sent with each camera frame, e.g. if the analog capture rate is three times the camera capture rate there will be in average three analog samples sent with each frame. However because of the buffering in the analog board the number of samples sent with each marker frame can differ, but the total number of samples will be correct.

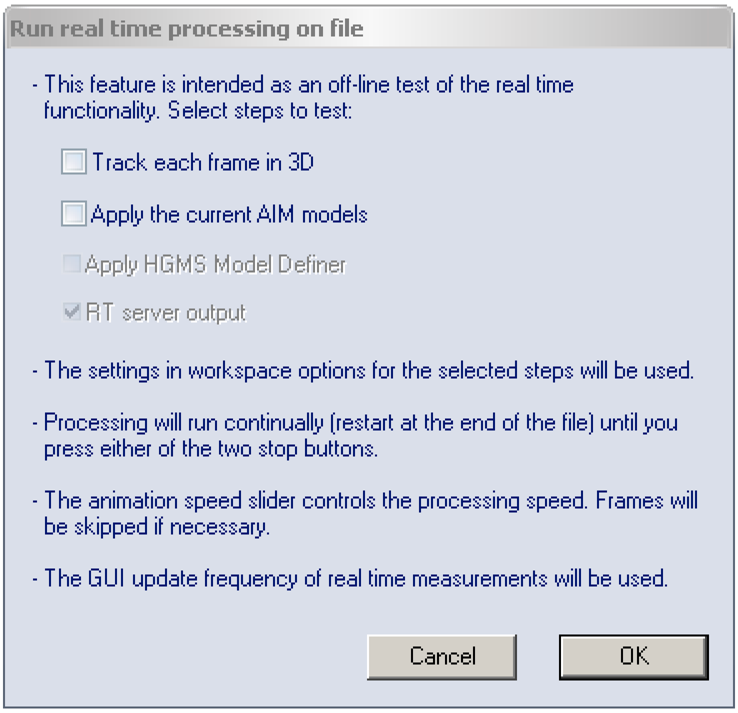
Check that the processing can be done fast enough in QTM by looking at the status bar, the RT frequency should be the same as the camera frequency. There will also be a message in the status bar if any RT frames are dropped or if there is a mismatch between the camera frames. For more information see chapter ”Main status bar”.

Use the following guidelines to get as fast real-time as possible:

- Activate only the necessary processing steps on the **Processing** page.
- Set the GUI update to 15 Hz on the **GUI** page, or shut it off completely to get the maximum performance.
- Check that the RT frequency is stable during the RT measurement. Lower the rate if it changes to much from the specified capture rate.

Real time processing on file

The real time can be tested from a tracked file with **Run real time processing on file**, this option is especially useful when testing the other program as it gives full control of what is sent.



When testing the RT output it is best to have an identified file and then uncheck all of the options in the **Run real time processing on file** dialog. Otherwise the data will be processed for each frame, which will change the data in the file. The options **Track each frame in 3D** and **Apply the current AIM models** should only be used if you specifically want to test these functions in real-time.

The speed of the real time process can be controlled with the **Playback speed** bar. Stop the real time with the **Stop** button on the **Playback** toolbar.

Outline of how to use real time

The following steps describe how to use real-time and send the data to another program, for example Visual3D.

1. Before you start the real-time, make sure that you have an AIM model or 6DOF bodies for the movement that you are going to capture, see chapter “Automatic Identification of Markers (AIM)” on page 302 respectively “6DOF tracking identifies rigid bodies from the 3D data of a measurement and calculates their translation and rotation with respects to the global coordinate system. It is activated with Calculate 6DOF on the Processing page in the Project options dialog and is controlled by the 6DOF tracking parameters, see chapter “6DOF tracking” on page 179.” on page 285.
   Note: It is best if the AIM model has been created on a measurement made on the subject that will be captured.

2. Switch on the camera system and start QTM.

3. Open a new file by clicking the **New file** icon.

4. Open **Project options** dialog and go to the **Processing** page.

5. Activate the **Real-time actions** that you want to use. For Visual3D the following actions should probably be used: **Track each frame: 3D**, **Apply the current AIM models** and **Calculate force data**.

6. Check the settings for the actions that have been selected.

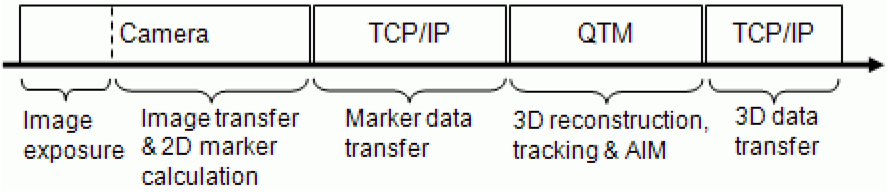
7. Go to the **Camera system** page and set the capture rate. The maximum capture rate depends a lot on the computer and the AIM model, but 100 Hz usually works with a dual-core computer.
8. Go to the **GUI** page and set the **Real time mode** screen update to 15 Hz. This step is not necessary, but it will give the computer more time to process the marker data.
9. Test the real-time with the motion that you want to capture. Look especially at how the AIM model is applied and if the RT frequency shown in the **Status** bar is close to the capture rate. If it differs too much, lower the capture rate, it might also help with a new AIM model or changing the tracking parameters.
    a. If the real-time is slow close all windows, including the **Data info** window and the **Trajectory info** windows, except for a **3D view** window.
    b. When the real-time is working fine you can even turn off the GUI with the **Disable GUI update** button GUI. This will reduce the processing capacity needed by QTM.
    c. AIM can be restarted with the **Apply AIM model** button.
10. When you are satisfied with the real-time in QTM, you can connect to QTM from the other program. That process is described below.

The following steps describe how to connect to another computer where a real-time process runs on QTM. For a description of the MotionBuilder plug-in see “Qualisys MotionBuilder Plugin” on page H - 1.

11. First you need to know the IP address of the measurement computer. On the computer with QTM, open **Network Connections** on the Windows **Start** menu.
    a. Right-click on the network that the other computer is connected to.
    b. Click on **Status** and then in the dialog click on **Support**. Write down the IP address.
12. Start Visual3D or any other program that you will use to retrieve the real-time data. To optimize the performance the other program must be started on another computer that is on the same network as the measurement computer.
    a. If there is no RT data in the other program try first to restart the RT in QTM and then close down all of the QTM windows and start QTM again.
13. In Visual3D follow these steps:
    a. Open a CMO file that includes a model file created from the current subject and with the same marker names as in the AIM model.
    b. Open the **Real-Time Capture** tab and connect to Qualisys_RT. Use the IP adress from step 1.
    The Qualisys Real Time Interface can be downloaded on the C-motion website.
    c. The subject should now be modelled in real-time.
    d. Click **Perform Capture** to capture the data.
    e. Click **Start buffering** to start and then **Stop** to stop the measurement.

Real time latency

Real time latency is the time it takes from the marker exposure to the when the TCP/IP signal can be received on another computer. The latency can be divided into the parts in the image below. The size of the different parts shows approximately how long time the different steps will take.



This delay will depend on the following factors:

Number of cameras
Increasing the number of cameras can increase the delay. Both because of the extra data and extra complexity in tracking

Computer
The computer performance will influence the latency and because QTM runs on Windows the latency may also differ depending on which other programs that are running.

Because the latency is system and computer dependent it is not possible to give a latency for any system. However for Oqus cameras there is a very elegant way of monitoring the system latency integrated in QTM. Enable the display of latency with the **Show Realtime Latency** option on the **GUI** page in the **Project options** dialog. When enabled the latency will continuously be updated in the status bar. The latency is calculated by comparing the time stamp at exposure with the time stamp of a UDP packet sent to the cameras by QTM at the end of the calculation pipeline. With this method the latency will match the latency of a third-party software receiving the data stream over UDP/IP from QTM’s realtime server. Contact Qualisys AB for if you want more help on how to measure the latency.

OSC integration

Open Sound Control (OSC) is a content format for messaging among computers and other multimedia devices. The format can be accessed through the RT output of QTM and any of the data in QTM (3D, 6DOF, 2D and analog) can be sent from QTM. For more information about the protocol and a demo program for Max contact Qualisys AB.

To record synchronized sound with the motion capture it is recommended to use the equipment described in chapter “Using synchronization box for SMPTE” on page 384.

Note: Each UDP packet can only contain 56 kByte of data. Therefore if the data can be truncated if you send a lot of data to the client for the OSC output.

Processing measurement data

Load project settings and calibration

Opening a project or restoring project settings

When opening a project or restoring project settings that have been saved as a backup, all of the settings in the **Project options** are changed. The only exception is the calibration. QTM will use the latest calibration made on the computer, that was made with the same cameras (placed in the same order). It means that the calibration may be loaded from another project when switching projects, it is then copied to the Calibrations folder in the project folder.

The calibration is also checked if the cameras are changed at **Locate** or **New**. If there is a matching calibration for the new cameras it will be loaded in the project.

Loading an old workspace file

The workspace files saved in QTM 2.5 or earlier can still be loaded in a project, it will then replace the current project settings. To load a workspace click on **Import**/**Load old workspace file** on the **File** menu. The data that is loaded are the settings in the **Project options** dialog and the window layouts.

QTM will use the latest calibration made on the computer, that was made with the same cameras (placed in the same order). It means that the calibration may be loaded from another project when switching projects, it is then copied to the Calibrations folder in the project folder

Loading a saved calibration

An old calibration can be loaded in QTM, so that files can be reprocessed with that calibration. However it is not recommended to use them when capturing a file as the cameras might have moved since the calibration was made.

The calibration can be loaded to the workspace or during the reprocessing of a file. Click on **Load other** on the **Current calibration** respectively the **Calibration** page. The following data is loaded with the calibration.

1. Current calibration and its **Transformation** settings.
2. The camera system and the camera order is loaded, so if the current cameras differ then you will lose the calibration when you start a new measurements.

3. The linearization files are loaded. But if you have newer linearizations stored in the camera, then you will get a message when you start a new measurement.

Data processing steps

Introduction to data processing

After the capture is finished the data must be processed in QTM to get 3D data, 6DOF data and force data. Some functions in QTM, such as AIM and exports, can also be performed in processing. The processing can be done automatically after the capture, manually in the current capture file or automatically on saved files by using**Batch processing**.

The settings of the automatic processing, which are the same for the processing directly after capture and batch processing, are specified on the **Processing** page in the **Project options** dialog. The only exception is the **Auto backup** option which is only available when making a measurement.

The following processing steps are available on the page:

1. **Auto backup**, see chapter “Auto backup” on page 249.
2. **Track the measurement**, see chapter “Tracking the measurements” on page 281.
    a. **3D**, see chapter “3D tracking measurements” on page 283.
    b. **2D**, see chapter “2D tracking of data” on page 298.
3. **Gap-fill the gaps**, see chapter “Gap fill trajectories” on page 72.
4. **Apply the current AIM model**, see chapter “Automatic Identification of Markers (AIM)” on page 302.
5. **Calculate 6DOF**, see chapter “6DOF tracking identifies rigid bodies from the 3D data of a measurement and calculates their translation and rotation with respects to the global coordinate system. It is activated with Calculate 6DOF on the Processing page in the Project options dialog and is controlled by the 6DOF tracking parameters, see chapter “6DOF tracking” on page 179.” on page 285.
6. **Calculate force data**, see chapter “Force data calculation” on page 311.
7. **Export to TSV**, see chapter “Export to TSV format” on page 314.
8. **Export to C3D format**, see chapter “Export to C3D format” on page 320.
9. **Export directly into Matlab**, see chapter “Export directly into Matlab” on page 326.
10. **Export to Matlab file**, see chapter “Export to MAT format” on page 322.
11. **Export to DIFF format**, see chapter “Export to DIFF format” on page 321.

The settings of each processing step can be found in the tree to the left in thedialog, except for **Export directly into Matlab** which has no settings. For information about the settings see each respective chapter under “Project options” on page 107.

⚠ **Important**: The two tracking options under **Track the measurement** are mutually exclusive and can therefore not be used at the same time.

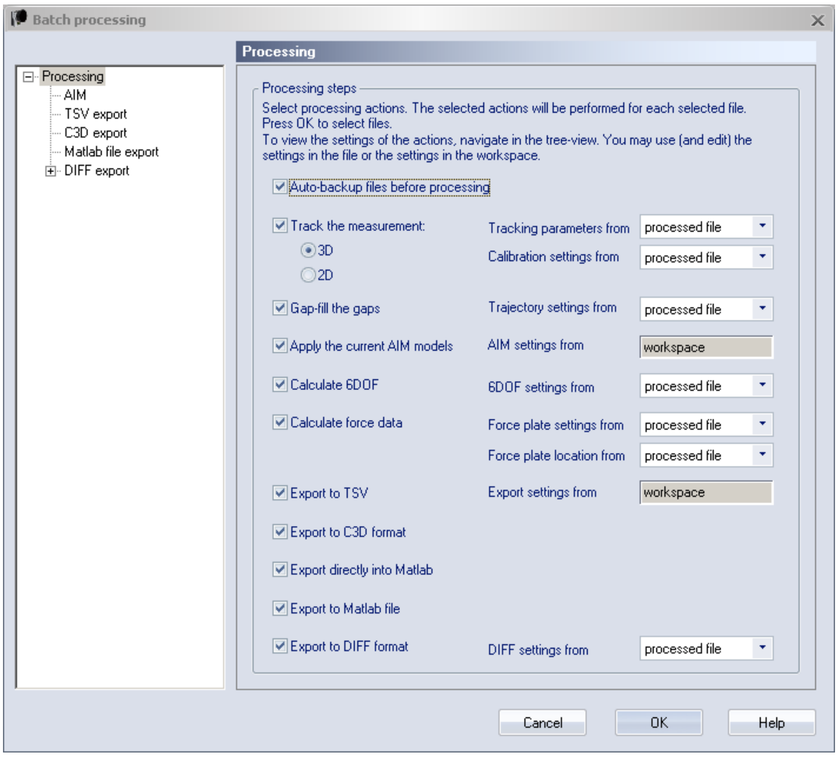
Batch processing

With batch processing, several capture files can be processed with the same settings. The same actions can be performed in batch processing as in the processing that is performed after a measurement has been captured.

Important: If **Track the measurement** is used when batch processing, the tracking of the saved files will be changed. This means that all of the trajectories will be unidentified again unless you apply AIM in the same batch process.

To start a batch process, follow the steps below.

1. Click **Batch process** on the **File** menu to open the **Batch processing** dialog.



2. Select the processing steps that you want to perform from the list. The processing steps are the same as those that can be applied in post-processing, see chapter “Processing” on page 170.

   Note: Use the **Auto backup files before processing** if you want to keep a backup of the file. The files are saved in the same directory with the current date and time in the filename.

3. Choose the source for the settings from the drop-down lists to the right. For some settings the only possible source is the workspace, for the other settings there can be three possible options:

   Processed file
   The settings used are the ones present in each processed file and cannot be edited. This option is always selected by default because the other options remove the original settings from each file.

   Project
   The settings are copied from the current workspace to each processed file. The settings can be edited in the tree-view to the left. Editing the settings will change the current workspace settings as well. This option

is often the best to use when you want to change the settings of the processed files.

Present file
The settings are copied from the present file (the file open in the QTM window where you opened the **Batch processing** dialog) and can be edited in the tree-view to the left. Only available for **Calculate 6DOF**, **Calculate force data** and **Export to DIFF format**.

Note: For **Track the measurement** the settings are split into **Tracking parameters** and **Calibration settings** so that the sources of these settings can be set separately. Make sure that the calibration is valid for all of the files you want to batch process when using the option **workspace**.

Note: For **Calculate force data** the settings are split into **Force plate settings** and **Force plate location** so that the sources of these settings can be set separately. This can for example be used on files where the location varies, but you want to process the files with the same **Force plate settings**. In this case, select **processed file** as the source of **Force plate location** and select **workspace** as the source of the **Force plate settings**. Edit the force plate settings by selecting them in the tree on the left.

4. Click **OK**. Select the files and click **Open**. The actions are performed on the files and then the files are saved and closed.
   Note: Any changes to a capture file are saved automatically, i.e. it will be saved if any step except the export options are used.

TSV (3D) and QTrac files can also be processed with batch processing. The table below shows which processing steps that can be performed.

Processing step	**TSV files**	**QTrac files**
Tracking	No	Yes, if Cap files.
Gap fill	No	Yes, if Seg files.
Apply AIM	No	Yes, if Seg files.
Calculate force	Yes, if the files have analog data and the settings of the force plate has been set.	Yes, if there are analog data and the settings of the force plate has been set.
Calculate 6DOF	Yes	Yes
Export to TSV	Yes	Yes
Export to C3D	Yes	Yes
Directly to Matlab	Yes	Yes
Export to Matlab file	Yes	Yes
Export to DIFF format	Yes, but the settings must be specified correctly.	Yes, but the settings must be specified correctly.

Tracking the measurements

Introduction to tracking

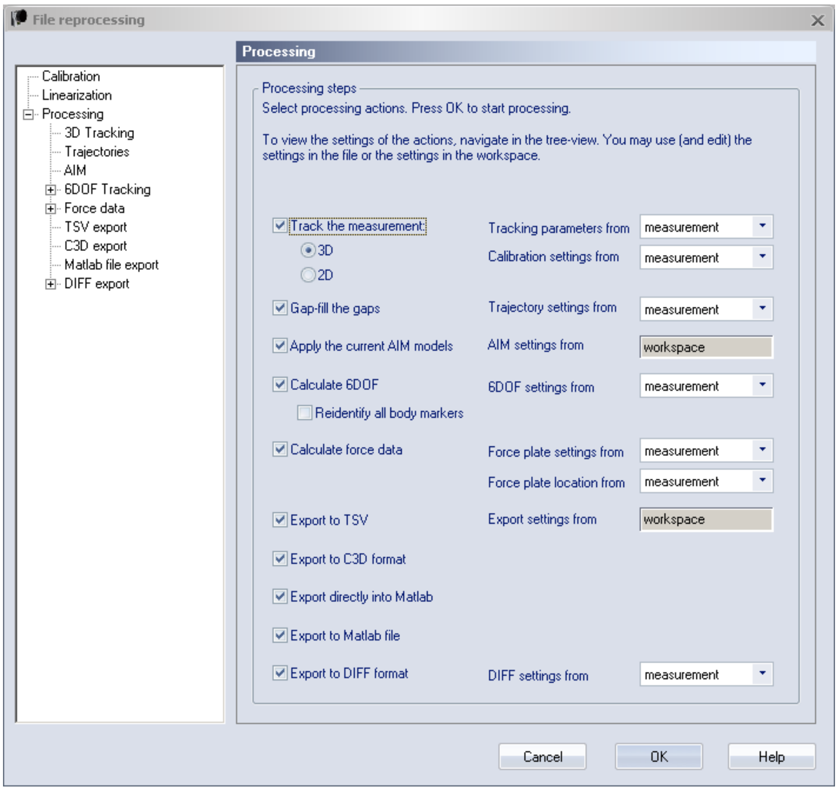
Tracking is the process which identifies the 2D data and displays it as trajectories bodies in a 3D view. QTM has two tracking options: 3D and 2D. The tracker is activated with **Track the measurement** on the **Processing** page in the **Project options** dialog. Then you have to choose one of the tracking options, you can only use one at a time.

The tracking can be changed in a saved file either by using batch processing or by reprocessing the file. However when changing the tracking some settings may be lost, see the respective chapters for more information.

Reprocessing a file

If there are problems with the tracking of a measurement or if you want to change the type of tracking, it can be reprocessed with other **Tracking**, **Calibration** and **Linearization** parameters than during the capture. This will, however, delete any other processing that has been made to the file, e.g. all trajectories will be returned to the **Unidentified trajectories** window.

Reprocessing is started by clicking the **Reprocessing** icon or **Reprocess** in the **Capture** menu. The reprocessing process is controlled from the **File reprocessing** dialog.



The **File reprocessing** dialog contains all of the settings from the **Project options** dialog that can be applied to a file in post-processing, see chapter “Processing” on page 170. Follow these steps to set the reprocessing steps.

1. Select the reprocessing steps that you want to perform from the list.
2. Choose the source for the settings from the drop-down lists to the right. For some settings the only possible source is workspace, for the other settings there are two possible options:

    Measurement
    The settings are copied from the file and can then be edited. This option will always be selected by default because the other option removes the original settings from the file.

    Project
    The settings are copied from the workspace and can be edited in the tree-view to the left. Editing the settings will change the current workspace settings as well.

    Note: When **Track the measurement** with **3D** or **2D** is used all of the trajectories will be unidentified again unless you use AIM in the reprocessing.

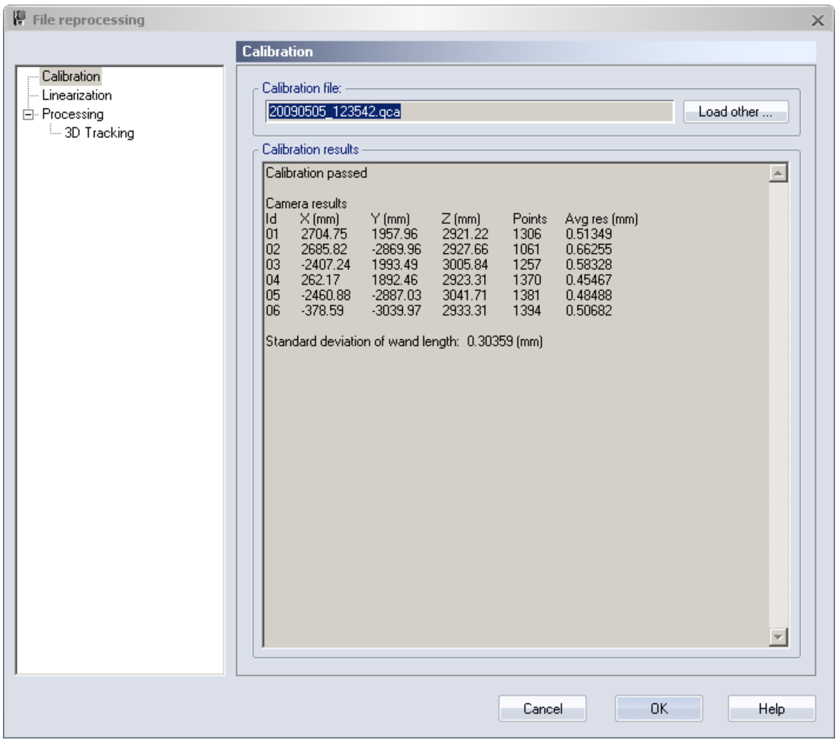
    Note: The **Track the measurement** settings are split into **Tracking parameters** and **Calibration settings** so that the source can be set separately. Make sure that the calibration is correct when using the option **workspace**.

    Note: The **Calculate force data** settings are split into **Force plate settings** and **Force plate location** so that the source can be set separately.
3. Click **OK** to start the reprocessing.

The **Calibration** and **Linearization** pages are connected for files that have been 3D tracked. This means that you have to change the calibration to change the linearization. The **Linearization** page is only displayed so that you can check that the linearization files are correct. However for a 2D tracked file there is no **Calibration** page and you can change the linearization on the **Linearization** page, see chapter “Linearization” on page 116.

Changing calibration

The calibration of the measurement can be changed in the following ways: by reprocessing the current calibration of the file or by replacing the calibration with another calibration with the same camera setup. For information on how to reprocess calibrations see chapter “Reprocess calibration” on page 244. How to change the calibration file in reprocessing is described below.



The calibration of a capture file is changed on the **Calibration** page in the **File reprocessing** dialog. The capture file's current calibration is shown under the **Calibration file** heading and the results are displayed under the **Calibration results** heading. Replace the current file with the wanted calibration file by clicking **Load other** and locating the file. If the current calibration file has been reprocessed it must opened again so that its parameters are reloaded, because otherwise the reprocessing process will run with the calibration results that are saved in the capture file.

Note: The calibration files are located in the **Calibration** folder in the project folder and the name of a calibration file includes the date when the file was made

To start reprocessing the file click **OK**. The new settings in the **File reprocessing** dialog are only valid for the active file. To keep the changes the file must be saved.

3D tracking measurements

3D tracking is the process which constructs 3D points from the 2D rays of the capture and then sorts the points into trajectories. It is activated with **Track the measurement** and the **3D** option on the **Processing** page and is controlled by the tracking parameters, see chapter “3D Tracker parameters” on page 172. For an example on how to test that the tracking is working properly, see chapter “3D tracking test” on next page.

Track the measurement with the **3D** option is used by default, but if it has not been used or if there are some errors in the tracking of the measurement can be reprocessed, see chapter “Reprocessing a file” on page 281. For information about how to correct errors in the tracking process see chapter “Troubleshooting tracking” on page 393.

Advice on how to set 3D Tracker parameters

The following list of advice is not comprehensive. There are cases where you might need another approach to how to set the 3D tracking parameters. The complete description of the four different tracking parameters can be found in the chapter “3D Tracker parameters” on page 172.

Prediction error

The **Prediction error** is by default set to 30 mm. Often this parameter can be lowered to about 25 mm to avoid that two trajectories switch place. How low it can be depends a lot on how far the trajectory is moving between two frames. If the trajectories are moving less than about 10 cm between two frames it can be even lower than 25 mm. If the captured motion includes very fast movements, e.g. a golf swing, it can help to increase this parameter to more than 150.

Max residual

The **Max residual** is by default set to 10 mm. Often it can be as low as 5 mm with the new linearization. One way to estimate the **Max residual** is to track the file and then plot the residual of all of the trajectories. Then you can set the parameter two or three mm higher than the maximum residual of the trajectories.

3D tracking test

Before the actual measurements, it is a good idea to test that the tracking is working properly for the current camera setup. Place all markers required on the measurement subject and also one marker in each corner of the measurement volume. Then perform the following test:

1. Open a new QTM file and perform a capture. The measurement should be the same as that which will be measured with the system.
2. Go to the **View** menu and open the **File information** dialog. In the dialog there is a list of the tracking residuals per camera. There are two numbers per camera. **Points** are the number of 2D points from the camera that are used to calculate 3D in the measurement. **Avg res** is the average residual for the 2D rays from the camera at the average marker distance in the whole measurement.

   Tracking residuals per camera at 6 m

Id	Points	Avg res(mm)
1	52626	0.8
2	44855	1.1
3	51045	1.3
4	47352	1.4
5	47197	0.8
6	48449	0.9
7	46879	0.9
8	43067	0.8
9	43570	0.8
10	49224	0.7

   Check that none of the cameras have a lot less **Points** than the others. If there is such a camera it either cannot see the markers or has moved since the calibration.

   Check that the **Average residual** are similar for the different cameras. If there is a camera that has a much higher residual the system needs a new calibration.
3. Check the number of trajectories in the **Unidentified trajectories** window. Each of the trajectories should represent a marker and therefore the number of trajectories should match the number of markers in the measurement. However, if some markers have been obscured during the measurement there will be some extra trajectories.
4. Check that the markers can be seen in the entire measurement volume by comparing the trajectories data to the four markers placed in the corners of the measurement volume. It is easier to see if the volume is viewed from above (XY- view).

If the tracking test is not acceptable check the following factors:

- Check calibration, a faulty calibration gives erroneous data to the tracking function, see chapter "Calibration of the camera system" on page 235.
- Check camera positioning, incorrect positioning results in hidden markers and problems with the calculation of markers; see chapter "Camera positioning" on page 250.
  Note: The system must be calibrated after the rearrangement.
- Check the tracking parameters; see chapter "3D Tracker parameters" on page 172.
  E.g. if the **Maximum residual** is too small one marker might result in several false markers.
- Check the size of the reflective markers; see chapter "Marker size" on page 254.
  E.g. if they are not visible in 2D views, check the focus or choose a larger marker or open the aperture a bit more (this requires a new calibration). If they are merging into one marker in 2D views, choose a smaller marker or close the aperture a bit (this requires a new calibration).

Tracking and accuracy

There is a connection between the tracking parameters and the accuracy. However as long as there are few gaps in the measurements you usually do not have to worry about this dependency.

Both setting the **Prediction error** and the **Max residual** too low and too high have a negative impact on the accuracy. If the options are set too low camera data can be discarded that should have been used for calculation of a trajectory, which can result in extra noise and gaps. If they are set too high the risk increases for false data to be used with the trajectory, which can result in increased noise. However, the likelihood for this is reduced because of the outlier filtering process in the 3D calculation, so it is better to set the parameters somewhat higher than expected.

Setting the **Prediction error** too high can also increase trajectory flipping, i.e. when two trajectories flip identity and continue in the other's track.

6DOF tracking identifies rigid bodies from the 3D data of a measurement and calculates their translation and rotation with respects to the global coordinate system. It is activated with **Calculate 6DOF** on the **Processing** page in the **Project options** dialog and is controlled by the 6DOF tracking parameters, see chapter "6DOF tracking" on page 179.

6DOF versus 3D

A single point in the mathematical sense can be fully described by its three coordinates (X, Y, Z) in a Cartesian coordinate system (3D). This can also be described as having three degrees of freedom. A physical body, such as a ship or an aircraft, requires three additional degrees of freedom to fully characterize its location, i.e. three rotation angles. The characterization of an object by three positional coordinates and three rotational angles is therefore called "six degrees of freedom" (6DOF).

To measure the 6DOF data of a physical body QTM uses 6DOF bodies (rigid bodies). A 6DOF body consists of markers placed on an object so that their mutual distances are constant. The physical design of a 6DOF body is described in the next chapter.

The 6DOF bodies must then be defined on the **6DOF bodies** page in the **Project options** dialog, see "Definition of 6DOF bodies" on page 287. Given the information on the **6DOF bodies** page the 6DOF tracking function can calculate the rotation and position of objects in the measurement.

Creating 6DOF bodies

The following three chapters describe how to create and define 6DOF bodies in QTM. First the 6DOF body must be designed according to the instructions in the next chapter. For the 6DOF tracking to be able to find measured 6DOF bodies (rigid bodies), the rigid bodies must have been defined on the **6DOF bodies** page in the **Project options** dialog. The definition of a rigid body consists of points located in a local coordinate system. Given this information the 6DOF tracking function can calculate the rotation and position of objects in the measurement.

How to design a 6DOF body

A rigid body is a configuration of points with fixed separations. For the 6DOF tracking function to work the rigid body must be defined by at least four points, which cannot all be in the same plane. There are at least five points that must be considered when designing a 6DOF body.

Active vs. Passive markers

In marine applications it is often best to use Long range active markers in the 6DOF body. Since the measurement volume often is too large for passive markers. However, passive markers can be used at shorter distances, check the marker size according to point below.

Marker size (passive markers)

The marker size of passive markers on the 6DOF body depends on the measurement volume. An estimation that can be used for the marker size in relation to the measurement volume is that the 2D data **xSize** and **ySize** in the **Data info** window should be at least 7. However, it can be smaller but then the residual increases.

6DOF body size

Another factor to consider is the size of the 6DOF body in relation to the marker size. Very large markers in small bodies increase the risk of merging markers (two markers appearing as one in the **2D view** window). Merging should be avoided as mush as possible. Try to design the 6DOF body so that it can be viewed by the camera system in most angles that will be used in the measurement without merging.

Marker distance

Try to keep the marker distance as large as possible, provided that they fit within the measurement volume (including the model’s movements). Ideally the distances between all pairs of points that can be chosen from every rigid body definition in a motion capture should be sufficiently distinct, i.e. differ by at least twice the **Bone length tolerance**. This will increase the accuracy of the angle calculations. It also makes it easier to avoid merging markers.

Marker configuration

There are two requirements that must be fulfilled by the 6DOF body for the 6DOF tracking to work.

a. The 6DOF body must consist of at least 3 markers.

Depending on the accuracy of the system, the point configurations of the 6DOF body definition may have to be chosen with some care. There are two situations that should be avoided.

b. Symmetric configurations of the points in the 6DOF body. Because the 6DOF tracking function can only identify rigid bodies if the distances from one point to every other point are unique. This is especially important when the 6DOF body only has 3 markers.

c. Similar configurations in different 6DOF bodies. Because in a measurement each definition should correspond to just one measured 6DOF body.

It is also important to consider the placement of the 6DOF body on the model.

d. Put the markers sufficiently high so that they are not obscured by the model when it rolls away from the camera.

e. Preferably, have the markers of the rigid body on a plate or similar, that fits securely to the model in a well-defined position. The position of the Centre of Gravity (COG) in relation to one of the markers should be known if you want to place the local coordinate system in COG.

The following two points are only tips to make it easier to define a horizontal xy plane for the local coordinate system.

f. Keep at least 3 of the markers on the same vertical level. In this way you can use the **Define x y plane with 3 points in body** option in the **Rotate body** dialog to get a defined xy- plane which is horizontal, see chapter “Rotate body” on page 183.

g. The line between the two markers farthest apart should be parallel with the longitudinal (roll) axis of the model. Together with the previous point this will mean that the initial position of the local coordinate system is horizontal and that the local x-axis is parallel with the global x-axis. The y-axis is always placed in the direction of the third point used in **Define x y plane with 3 points in body**.

Definition of 6DOF bodies

When the 6DOF body has been designed according to the previous chapter you must define the points of the body in QTM. The following four ways can be used to define 6DOF body definitions in the list on the **6DOF tracking** page:

1. Load a saved file with 6DOF body definitions with the **Load bodies** option on the **6DOF bodies** page.
   Note: This will delete any other 6DOF definition in the list.
2. Import the points from a measurement of a stationary object. Use the **Define 6DOF body** function on the **Trajectory info window** menu or **Import markers** on the **6DOF bodies** page.
   Note: The bodies created with the **Define 6DOF body** option is added to both of the **Rigid bodies** lists in the **Project options** dialog and in the current file. The 6DOF data is then calculated in the file, which means that the 6DOF data of all other bodies in the file will be updated as well.
3. Define the points of the body directly on the **6DOF bodies** page, see chapter “Rigid bodies” on page 180.

4. Acquire the points through a capture in preview mode, see chapter “Acquire body” on page 185. This method is described in chapter “Outline of how to define 6DOF bodies” below.

Note: Make sure that at least three points are included in the 6DOF body definition.

The points in the 6DOF body can be renamed by double-clicking on the point on the **6DOF bodies** page. If the trajectory has been labeled before creating the 6DOF body, then the point will keep that name.

When you have defined a 6DOF body in QTM it is a good idea to save the definition with **Save bodies** on the **6DOF tracking** page. This way you can always retrieve the bodies if you need to restore the settings after processing the file.

The same markers can be used for several bodies. The benefit of this feature is that each body can have different local coordinate system definitions, which means different rotation and translation. When using this feature the bodies must be created with the following two prerequisites.

5. The bodies must use exactly the same markers. with the same names, in its respective definition.
6. The bodies may however have different names.

Outline of how to define 6DOF bodies

When capturing 6DOF data it is important to define the correct 6DOF bodies in QTM before starting the capture. The definition can be done in four ways: manually, with a capture, from a file or load a previously saved 6DOF body. In the steps below it is just described how to acquire the points through a capture in preview mode. For more information about the definition of 6DOF bodies see chapter “Definition of 6DOF bodies” on previous page.

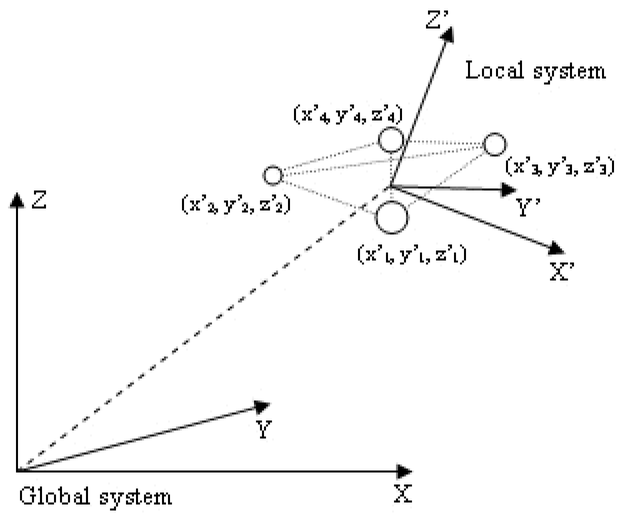
Note: To get help on how to design a 6DOF body see chapter “How to design a 6DOF body” on page 286.

Follow these steps to acquire the 6DOF bodies with a 3D capture:

1. Switch on the camera system.
2. Start QTM and open a new file by clicking the **New file** icon .
3. Place the rigid body in the measurement volume so that the position of the desired local coordinate system is known in reference to the global coordinate system. One way is to place the body so that the desired local coordinate system is aligned with the global coordinate system and then the local origin can just be translated to the desired position.
4. Check that the markers on the 6DOF body do not merge in any of the cameras' 2D views.
5. The following three steps are not needed to acquire the data, but it makes it possible to preview the changes made to the body without closing the **Project options** dialog.
    a. Activate **Track each frame in 3D** and **Calculate 6DOF** for the **Real time actions** on the **Processing** page. Click **OK**.
    b. Change to a **3D view** window.
    c. Open the **Data info** window and change the display to **6DOF data**.
6. Open the **Project options** dialog in QTM and go to the **6DOF tracking** page.
7. Click**Acquire body**.

8. Click **Acquire**. When the collection is done the new 6DOF body definition will appear on the **6DOF tracking** page.
9. If you want to change the local coordinate system follow these steps:
    a. Go to the **6DOF Tracking** page and rotate and translate the local coordinate system to the correct orientation, see chapter "Rotate body" on page 183 and "Translate body" on page 182.
    b. Click **Apply** to see the rotation and translation in the **3D view** window.
10. If you want to change the coordinate system that the 6DOF body refer to follow these steps:
    a. Double-click on **Global origin** to open the **Coordinate system for rigid body data** dialog where you can change the origin and the rotation of the coordinate system, see chapter "Coordinate system for rigid body data" on page 184.
    b. Click **Apply** to see the changes in the **Data info** window.
11. Repeat the steps 3 – 10 for all of the 6DOF bodies that will be included in the measurement. Just one rigid body at a time can be acquired with **Acquire**.
12. You can save your 6DOF bodies with **Save bodies** and then load them another time with **Load bodies**.
    Note: **Load bodies** will overwrite any other bodies in the list.
13. Make sure that **Calculate 6DOF** is activated for the **Capture actions** on the **Processing** page, otherwise the 6DOF bodies will not be tracked in the captured file.
14. Click **OK** to exit the **Project options** dialog.

Definition of local coordinate system



When you have defined the points of the 6DOF body in QTM you can change the definitions of the local coordinate system. If you have used **Acquire body** or **Import markers** the local coordinate system is by default placed where the global coordinate system were when the points were captured.

The local coordinate system is used in the calculation of rotation and position of the measured rigid body in reference to a reference coordinate system. Therefore it is important that the local coordinate system is defined according to the specifications of the measurement. The local coordinate system should have an orientation and a location in reference to the points in the 6DOF body definition, which is well-defined. Use a definition where the normal orientation of the body is the same as no rotation, i.e. aligned with the reference coordinate system.

When you have decided where the local coordinate system should be, use the **Translate** and **Rotate** functions on the **6DOF Tracking** page to specify the local coordinate system, see chapter “Rigid bodies” on page 180.

Then you should also decide which coordinate system that the 6DOF body data should refer to. This done in the **Coordinate system for rigid body data** dialog, which is opened by double-clicking on **Global origin** on the **6DOF bodies** page. See chapter “Coordinate system for rigid body data” on page 184 for the alternatives.

Note: If you want to change these setting in a capture file you must reprocess the file with the new setting.

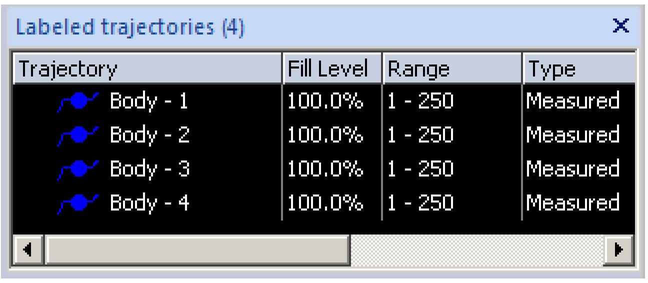
Tracking 6DOF bodies

To track the 6DOF data you must 3D track a file and then activate **Calculate 6DOF** on the **Processing** page in the **Project options** dialog. The 6DOF tracking function can either be applied in the processing directly after a motion capture or by reprocessing a 3D tracked file. You can also use reprocessing to edit the 6DOF data, see chapter “Calculating 6DOF data” below.

QTM will identify the trajectories of the 6DOF bodies and place them in the **Labeled trajectories** window. For each 6DOF body the trajectories are named the same as the body with a number at the end, see image below. The color of the trajectories will be slightly brighter than the 6DOF body color.

If the bodies cannot be tracked in a file, you will get a warning with how many bodies that have failed. There is also a warning if a label is repeated in the **Labeled trajectories** list, in which case only the first label is used by the 6DOF calculation.

Note: If several bodies share the same marker definition and name, the trajectories will only be shown once in the **Labeled trajectories** window.



Calculating 6DOF data

When 6DOF tracking is activated, QTM attempts to identify the measured rigid bodies from the 3D data. It is done by comparing the distances between points in the 6DOF body definitions to all distances between the computed markers’ locations. The following rules are used for calculation of 6DOF data.

- For a marker to be included in the 6DOF body none of the distances to the other markers in the body may exceed the **Bone length tolerance**.
- When the trajectories have been identified the tracker computes the location and orientation of each measured rigid body in the coordinate system of the motion capture. The effect on the 6DOF data of the **Bone length tolerance** differs slightly between RT and in files.

- In RT the marker that is outside the tolerance will be unidentified and the 6DOF body will be calculated from the remaining markers.
- In a file the automatic 6DOF tracker will discard the whole trajectory that is wrong and then calculate the 6DOF body from the other trajectories. However if you identify the trajectories manually and then just **Calculate 6DOF**, then there will be no 6DOF data in frames where a marker is outside the tolerance.
- If there are frames during the capture when there are less than three markers that can be tracked on the 6DOF body, the 6DOF tracking function cannot calculate the 6DOF body in those frames.
- Each 6DOF body definition can just have one solution in the measurement, but the 3D trajectory can be included in several 6DOF bodies as long as the 6DOF definition is the same and the body points have the same names.

The 6DOF data can be edited in a capture file. This is because each 6DOF body consists of labeled trajectories, from which the 6DOF data is calculated. This means that if the 6DOF data is wrong you can look at the 3D data to find the problem. To edit the 6DOF data just change the 3D data of the labeled trajectories and then reprocess the file with only the **Calculate 6DOF** option activated. Then QTM will recalculate the 6DOF data from the available 3D data.

When reprocessing a file the 6DOF data is reprocessed in the following ways depending on the processing steps.

Calculate 6DOF
The descriptions are for **Calculate 6DOF** without the **Reidentify all body markers** option. With that option the reprocessing will try to reidentify all of the markers in the 6DOF bodies. Unless the labels are also included in an AIM model.

Data in all of the 6DOF labels
When there are data in all of the labeled trajectories for the 6DOF body, then the 6DOF data is calculated directly from that data. QTM does not try to identify any other trajectories.

At least 3 6DOF labels with data
If at least 3 6DOF labels contains data then QTM will calculate the 6DOF data from those and not try to identify the rest of the labels

Less than 3 6DOF labels with data
When less than 3 labels in the 6DOF body is empty, then QTM will try to identify the 6DOF body again. Which means that the identified trajectories will be unidentified and the body calculated from scratch. This also means that trajectories will be placed last in the **Labeled trajectory info** window.

Apply the current AIM models

6DOF body not included in the AIM model
The 6DOF bodies that are separate from the AIM model will not change at all, unless **Calculate 6DOF** is applied as well.

6DOF body included in the AIM model
6DOF bodies that are included in the AIM model will be reidentified. However the 6DOF data is not recalculated unless the processing step **Calculate 6DOF** is activated.

Virtual markers calculated from 6DOF data

The 6DOF bodies can be used to calculate virtual markers in relation to the 6DOF position. These markers are calculated both in real-time and in a file, and can then be exported and analyzed like any other marker. To virtual markers have the type **Virtual** in the **Trajectory info** window and the residual is always 0. The trace of a virtual marker is a dotted line just like trace of a gap-filled part.

There are two different ways of creating virtual markers:

Virtual point in 6DOF body definition
The virtual trajectory of the virtual point is calculated as soon as the 6DOF body has data, i.e. there must be at least 3 real markers in a frame to calculate the virtual marker. To make a point virtual in the 6DOF body definition you just select the checkbox in the **Virtual** column of the **Rigid bodies** list, see chapter “Rigid bodies” on page 180.
To get the desired virtual position you can either put a temporary marker in the correct position when creating the body and then remove the marker when making the measurement. Or you can add the point with the **Add Point** option on the **6DOF bodies** page and enter its position manually.

Virtual trajectory part in gaps of 6DOF marker
The virtual trajectory part is calculated when a real marker is lost and the **Calculate missing markers in rigid bodies** option on the **6DOF bodies** page is activated. There must be at least 3 real markers left in the 6DOF body to calculate the virtual marker in a frame.

Rotation angles in QTM

The rotation angles of a rigid body can be defined in any number of ways, see chapter “Euler angles” on page 186. The **Qualisys standard** of the rotation angles are defined as:

- Rotation around the X-axis is called roll.
- Rotation around the Y-axis is called pitch.
- Rotation around the Z-axis is called yaw.
- Positive rotation is defined as clockwise rotation when looking in the direction of the axis
- The angles are applied to the local coordinate system in the order: roll, pitch and finally yaw. Therefore to find the rotation of a rigid body with given roll, pitch and yaw angles from QTM, apply the rotations in the same order: first roll, then pitch and finally yaw.
- QTM uses the following default ranges for the angles, see also the figures below:

  $-180° \leq \text{Roll} \leq 180°$

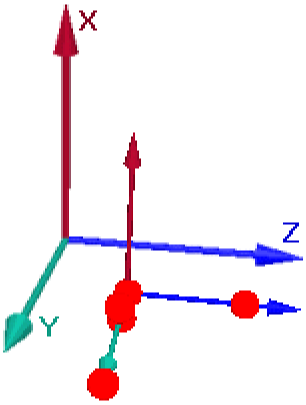
  $-90° \leq \text{Pitch} \leq 90°$

  $-180° \leq \text{Yaw} \leq 180°$

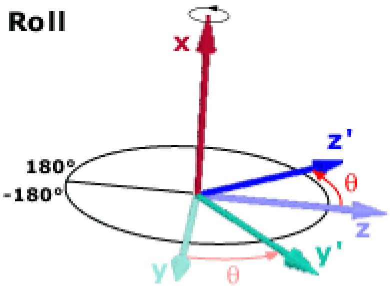
  In these ranges, roll, pitch and yaw are unambiguous and can describe any orientations of a rigid body.

Important: When the pitch (ϕ) is close to ±90°, small changes in the orientation of the measured rigid body can result in large differences in the rotations because of the singularity at ϕ=±90°, see “Rotation angle calculations in QTM” on page G - 1.

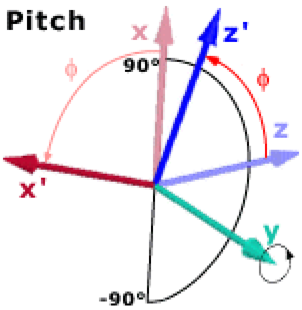
Below follows an example to show the definitions of the rotation angles. It starts with a 6DOF body, which is in alignment with the global coordinate system.



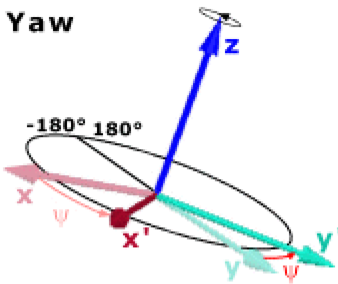
First the local coordinate system is rotated around the X-axis (roll) with an angle θ to the new positions y' and z' of the Y- and Z-axis.



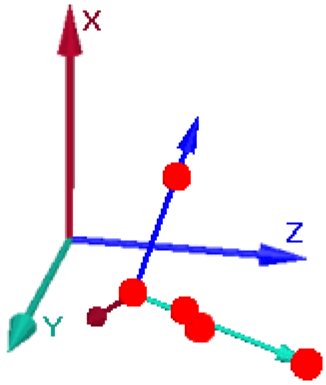
After the roll the local coordinate system rotates around the Y-axis (pitch) with the Y-axis in its new position. The X- and Z-axis is rotated with an angle ϕ to the new positions x' and z'.



Finally the local coordinate system is rotated around the Z-axis (yaw) with the Z-axis in its final position. The X- and Y-axis is rotated with an angle ψ to the new positions x' and y'.



After the rotations the rigid body has a new orientation in reference to the global coordinate system, see figure below.



Another description of rotations is to use the rotation matrix, which does not have a singularity. QTM uses the rotation matrix internally to describe the rotation of rigid bodies, and when exporting 6DOF to TSV files the rotation matrix is included for all bodies in all frames, together with roll, pitch and yaw angles. For a description of the calculation of the angles from the rotation matrix see “Rotation angle calculations in QTM” on page G - 1.

6DOF real-time output

With 6DOF real-time output you can export the 6DOF data to another computer. The data that can be exported are position, rotation angles, rotation matrix and residual. The 6DOF data will use the same definitions as you have specified in QTM and the 6DOF real-time output is accessed the same way as the 3D real-time, see chapter “Real time in QTM” on page 271.

Note: If you are using the old real-time called the Legacy RT server it must be activated on the **Legacy RT server** page in the **Project options** dialog. The Legacy RT server is accessed with a DLL file.

6DOF analog output

With the analog output option the information about 6DOF bodies’ positions and rotations can be used in feedback to an analog control system. The analog output is activated on the **6DOF Analog export** page in the **Project options** dialog and it is only used during measurements of 6DOF bodies. To be able to output the analog data the 6DOF body must be tracked, i.e. **Calculate 6DOF** must be activated and a **3D View** window must be open. The analog signal is sent whenever the 6DOF body is tracked, which means that the output will only be in sync with every captured frame in preview and in 6DOF real-time output.

Note: In regular capture it will only work when **Display the last fetched frame** is selected and then it will only be used on the frames that are tracked and displayed.

The data values that will be used are selected on the **6DOF Analog export** page, see chapter “6DOF analog export” on page 205. Since the required board, see chapter “Analog output hardware” on the facing page, has 16 channels the output is limited to 16 data values of 6DOF bodies. In order to maximize the use of the 16 bit resolution, the data on each channel can be scaled so that the resulting value is then converted to a voltage which represents the value’s proportional position within the range.

Note: The output of a channel will be 0 V if the body is not found. If the input value is outside of the input range the output will be either the **Output min** or the **Output max** value depending on the input value.

Analog output hardware

To enable analog output a D/A board must be installed in the measurement computer. Analog output is only available with the Measurement Computing board PCI-DAC6703. This is a 16 channel board with 16 bit resolution. To install the card insert it in any available PCI slot and then start the program Instacal from Measurement computing.

Examples of how to use 6DOF bodies

The chapters below contain examples of how to use 6DOF bodies in different ways and combinations.

How to use 6DOF bodies

The standard way to use the 6DOF bodies are to create a body for each separate object that you want to measure. In this case the markers on each subject have no relation to markers on the other subjects. Follow these instructions to use 6DOF bodies.

1. To create a 6DOF body follow for example the instructions in chapters “How to design a 6DOF body” on page 286 and “Outline of how to define 6DOF bodies” on page 288.
2. It is important to think about the definition of rotation and position of the local coordinate system, see chapter “Definition of local coordinate system” on page 289.
3. It is also important to consider which Euler angles definition you want to use, because the calculated angles will differ depending on the definition, see chapter “Definition of rotation axes” on page 188.
4. The points in the 6DOF definition can be renamed on the **6DOF bodies** page.
5. Remember to turn on the **Calculate 6DOF** option on the **Processing** page.

These are some tips on how you can improve the 6DOF data.

- If you are streaming the data in real time it can help to activate the **In real time...** setting on the 6DOF bodies page so that the identification starts even if a marker is hidden.
- The 6DOF data can sometimes be improved with marker filtering, see chapter “How to improve 6DOF data with marker filtering” on the facing page.
- A special case for 6DOF bodies is if you need two sets of 6DOF data from the same rigid body. This can be done by adding the body twice and make sure that the points have the exact same names. The name of the 6DOF body on the other hand can differ.

How to use 6DOF bodies in an AIM model

In this case the 6DOF bodies are placed on parts that move together, e.g. a human body. Then the best approach is to use AIM model to identify the markers and then calculate the 6DOF bodies from the already calculated markers.

1. Create the bodies and their definition in the same way as in chapter “How to use 6DOF bodies” above. However it is important that you name the points of the bodies in the same way as they are named in the AIM model.
2. Create an AIM model from the subject, see chapter “Generating an AIM model” on page 302. If you already have an AIM model with the correct marker setup you can add the current measurement to that AIM model.
   Note: The AIM model can contain markers that are not included in 6DOF bodies.
3. Activate both **Apply the current AIM model** and **Calculate 6DOF** on the **Processing** page. The AIM model will be applied first and then the 6DOF

bodies will be calculated from the identified labels.

Note: If AIM fails to identify a marker, the 6DOF calculation will not try to identify it either even if you select the **Reidentify all body markers** in reprocessing. In most cases the best way to fix this is to manually identify the data and add it to the AIM model.

4. There can still be 6DOF bodies in the file that are not included in the AIM models. These will only be identified and calculated when the **Calculate 6DOF** option is activated, i.e. if you reprocess the file and only apply the AIM model the trajectories of the separate bodies will not be reidentified.

How to use virtual markers in an AIM model

The virtual markers in the 6DOF functionality can be used in a regular AIM model if it contains markers that are actually rigid bodies. This example describes the case when the 6DOF data is only used to create virtual markers and therefore the actual 6DOF data is not really important.

1. Create an AIM model as described in chapter "Generating an AIM model" on page 302. If you want to use a completely virtual point then you can add that as an empty label in the AIM model.
2. Disable the display of the rigid bodies with the **Show rigid bodies** option on the **3D view settings** page. Otherwise you will see for example the coordinate systems in the 3D view window.
3. Increase the **Bone length tolerance** to 20 mm on the **6DOF Tracking** page. This will make sure that the 6DOF bodies can be calculated even if the marker positions is not completely rigid.
4. Also activate the option **Calculate missing markers in rigid bodies** on the same page so that the virtual markers are calculated when a real marker is lost.

   Note: There must be at least three real markers identified in a frame to calculate the virtual marker.
5. When you measure a new subject.
   a. Unless you are really sure that markers on the rigid body has not moved, then you have to delete all of the bodies in the **Rigid bodies** list.
   b. Create the 6DOF bodies from a static file of the subject where AIM has identified the markers. Select at least four markers that does not move in relation to each other, and then use the **Define rigid body** option in the **Trajectory info window** menu. The trajectories will keep the labels they got from the AIM model.

      Note: All of the real markers in the 6DOF body must be included the AIM model, the others will not be identified. However 6DOF bodies that are completely virtual will be calculated even if they are not in the AIM model.
6. Make sure that both **Apply the current AIM model** and **Calculate 6DOF** are activated on the **Processing** page.

How to improve 6DOF data with marker filtering

The marker filtering functionality can be used to improve the 6DOF data by correcting merged markers or by removing marker data that is not circular enough. There are three different settings that can be used to filter the data, which setting to use must be tested in your system. For more information about the settings see chapter “How to use circularity filtering and correction of merged markers” on page 267.

Correct merged markers
The corrected merged markers are always used by the 3D tracker, which in most cases will give you a more stable 3d position when two markers merge.
The markers marked as uncorrected non-circular are only used by the 3D tracker, if there are less than 2 cameras with circular markers.
The default **Circularity** setting is 50, which usually works well. The correction works better with larger markers with a 2D size of at least 320 or larger.

Correct merged markers, remove others
This is the same as the **Correct merged markers** setting, except that the uncorrected non-circular markers are not used at all by the 3D tracker.
Use this setting if you have markers with a 2D size of at least 500. Depending on the system the **Circularity** setting may also be lowered to between 10-20.

Remove all non-circular markers
The markers marked as non-circular are not used by the 3D tracker.
Just as with the **Correct merged markers, remove others** setting it works best with markers larger than 500 in 2D size. Because then you can also use a **Circularity** setting between 10 and 20.

It is not possible to say which setting is the best for any system. But in general the **Correct merged markers** works quite well. Especially since there is a filter in the 6DOF tracker which removes any 3D marker that is more than % outside the definition.

Note: If there are too many non-circular markers in the camera, it will stop sending data to QTM that can be corrected.

2D tracking of data

With the 2D tracker you can track the 2D data of one camera and present the markers as trajectories in the **3D view** window. Because just one camera is used to track the trajectories, they will all be in a plane in front of the camera. This means that you can only measure the distances in two dimensions.

As the 2D data is tracked only in a plane no calibration is used to track the data. The only settings that are needed is the distance between the plane and the camera. That setting and other 2D tracking settings are set on the **2D tracking** page in the **Project options** dialog, see chapter “2D tracking” on page 175.

Note: The 2D tracker can only track one camera at a time and it can only be used on a captured file and not in real-time.

2D tracking can be applied either after a measurement as a processing step or by reprocessing a file, see chapter “Reprocessing a file” on page 281. To use 2D tracking after a measurement you have to activate it on the **Processing** page in the **Project options** dialog. And then check the settings on the **2D tracking** page.

Using 2D tracking

When capturing data which will be used by the 2D tracker it is important that the movement only takes place in a plane and that the cameras are placed so that the camera sensor is parallel with the plane of the movement. Otherwise the distances in the measured by QTM will not be correct. After you have placed the cameras follow these steps to use 2D tracking.

1. Before you start the capture you should activate 2D tracking with **Track the measurement** and **2D** on the **Processing** page in the **Project options** dialog.

Then set the settings on the **2D tracking** page, see chapter "2D tracking" on page 175.

a. Choose which camera to use. 2D tracking can only be applied to one camera at a time. So if you have more than one camera in the camera system you must retrack the file to get the trajectories for another camera.

b. It is a good idea to use filter so that you do not get may short trajectories.

c. Measure the distance between the measurement plane and the camera sensor and enter the **Distance to measurement plane** setting.

d. Define the orientation of the coordinate system with the three settings for the axes.

2. Start a capture with **Start capture** on the **Capture** menu. 2D tracking cannot be applied in real-time so you have to make a measurement to see whether your settings are correct.
3. When the 2D tracker is finished the result is trajectories that can be processed and handled exactly as trajectories created by 3D tracking. The only difference is that the trajectories will all be in the same plane.

Identification of trajectories

Manual identification of trajectories

Even if you plan on using an AIM model to identify the trajectories, you need to identify at least one file by hand, since an identified file is needed to create an AIM model. Manual identification can be performed with any of the following three methods, and they can also be used together.

Quick identification method
Manual identification is best done with the quick identification method. Follow these steps to use this method:

1. To use the quick identification method you need to have a label list in the **Labeled trajectories** window. Load a label list or enter the labels manually with **Add new label** on the **Trajectory info window** menu.
2. Select the first label in the **Labeled trajectories** window that you want to identify.
3. Hold down Ctrl + Alt or select the **Quick identification** cursor  and click on the marker or trace in the **3D view** window that matches the selected label.
    a. When you click on the marker or trace, it will be added to the selected label. If you have not selected any label, the marker you click on will be added to the first empty label in the list. If there is no empty label in the list, the marker you click on will be added as a new label at the end of the list, and you can edit its name at once.
    b. You can also add additional parts to a trajectory with the quick identification method. Select the labeled trajectory in the list, then hold down Ctrl + Alt and click on the unidentified marker.
    However, if the two trajectories cannot be joined (because of too much overlap) the selection will just move to the next label without any action.
    c. By holding down Shift when clicking on the trajectory you can join a trajectory to the previous label in the **Labeled trajectories** window. This option can for example be used if the most recently identified trajectory only covers a part of the measurement range. In this case, you can add more parts by clicking on the next part while holding down Shift.
4. QTM will move the selection automatically to the next label in the list. Hold down Ctrl + Alt every time you click on the next corresponding marker. Continue until all markers in the **Labeled trajectories** window are identified.
5. You can start and restart the process on any label in the list.
    a. This means that if you click on the wrong marker, you can undo the identification with Ctrl + z and then make sure that the correct label is selected and use quick identification again.

Drag and drop method
Manual identification can also be performed by dragging and dropping trajectories. The following drag and drop methods are available to identify trajectories.

From the **Unidentified trajectories** window to the **Labeled trajectories** window.

From the **3D view** window to the **Labeled trajectories** window.

One trajectory (trace) in the **3D view** window to another trajectory (trace) in the **3D view** window.

A trajectory can be dropped either on a label to add the data to that label or in the empty part of the **Labeled trajectories** window to create a new label. If it is dropped on a label that already contains data, the two parts will be joined if they do not overlap too much.

Identify method
It is also possible to use the **Identify** option on the **Trajectory info window** menu, which appears whenever you right-click a trajectory, either in a 3D view window or in a Trajectory info window. For information about functions in the **Trajectory info** windows see chapter "Trajectory info window menu" on page 69.

There are two more features that are useful when identifying trajectories.

- The keyboard shortcut **c** can be used to center on the selected marker in a **3D view** window.
- The **Trajectory info window** menu option **Center trajectory in 3D**, will also center on the selected trajectory or part. However, if the trajectory is not visible at current frame it will also move the current frame to the first frame of the trajectory.

Tips and tricks for manual identification

The following chapter describes some different tips and tricks to help you in the manual identification of trajectories. However it is important to first use the AIM functionality because it will significantly reduce the identification time, see chapter "Automatic Identification of Markers (AIM)" on next page.

Sometimes the trajectories have to be manually identified, for example when identifying the file for the AIM model. Then you can use the following tips to make the process faster and easier.

Visualization tips:

1. Use different colors on the trajectories so that it is easier distinguish between them in the 3D view.
2. Use bones between markers so that the marker pattern is more visible.
3. View the trace of the trajectories to check which trajectories that match.
    a. How much trace you can show depends on the measurement, for example in a straight movement you can show the trace for the whole measurement, but if it is a long measurement with a rotation movement the trace have to be shortened so that you can still see the individual traces.
    b. Turn off the trace for some of the labeled trajectories if it is hard to see the individual traces.

Identification methods:

4. Use **Quick identification** to identify markers in the 3D view.
5. Drag and drop markers from the 3D view to the Labeled trajectories list or on to a trace in the 3D view.

6. If it is a long measurement start with identifying just a part of the measurement and then generate an AIM model and apply that to the whole measurement.

Other tips

7. Hover the mouse over a marker or trace in the 3D view to get information about the range that it covers to check where it can fit. When hovering over a marker you get the information for the whole trajectory, while hovering over a trace only gives the information for that part of the trajectory.
8. Use the **Center on trajectory** tool to center the view on the trajectory that you are interested in.
9. Use Alt-click to select only a part if parts have not been joined correctly. Then drag and drop it to where you want it.
10. If there are unwanted reflections that have been tracked in the file view the trace and select it with Shift + drag and then delete it. Then you can delete several trajectories at once, but if the reflection is at the exact same position every time you have to repeat the process several times because only the visible traces are selected.
11. Use the Cut trajectory trace tool to cut a trace and then use Alt-click and drag and drop to move the trace to another trajectory.

Automatic Identification of Markers (AIM)

Automatic identification of trajectories in the QTM software is performed by a module called AIM (Automatic Identification of Markers). The AIM model is created from identified files and can then be applied to any measurement that captures similar motions with the same marker set. This makes AIM very flexible and powerful - any motion can be made into an AIM model. More details about how AIM works can be found in chapter h“How AIM identifies trajectories” on page 307.

Generating an AIM model

AIM needs a model file to work, which is generated from a measurement or multiple measurements with identified trajectories.

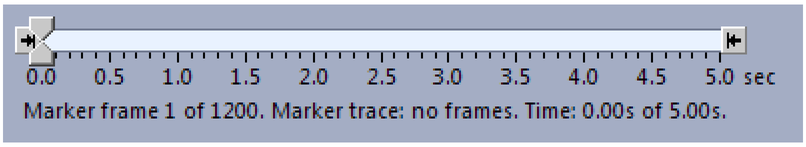
⚠ **IMPORTANT:** To get the best AIM model it is important that you continue to add QTM files with the same marker setup to the AIM model, if the automatic identification fails. You can add files with subjects of different sizes, as long as you have used the same method to place the markers. Use the **Add to existing model(s)** setting when you identified the file manually, because if you create a new model for each subject you will not use the full potential of the AIM functionality.

Follow the steps below to add a file to a model file or generate a new model file.

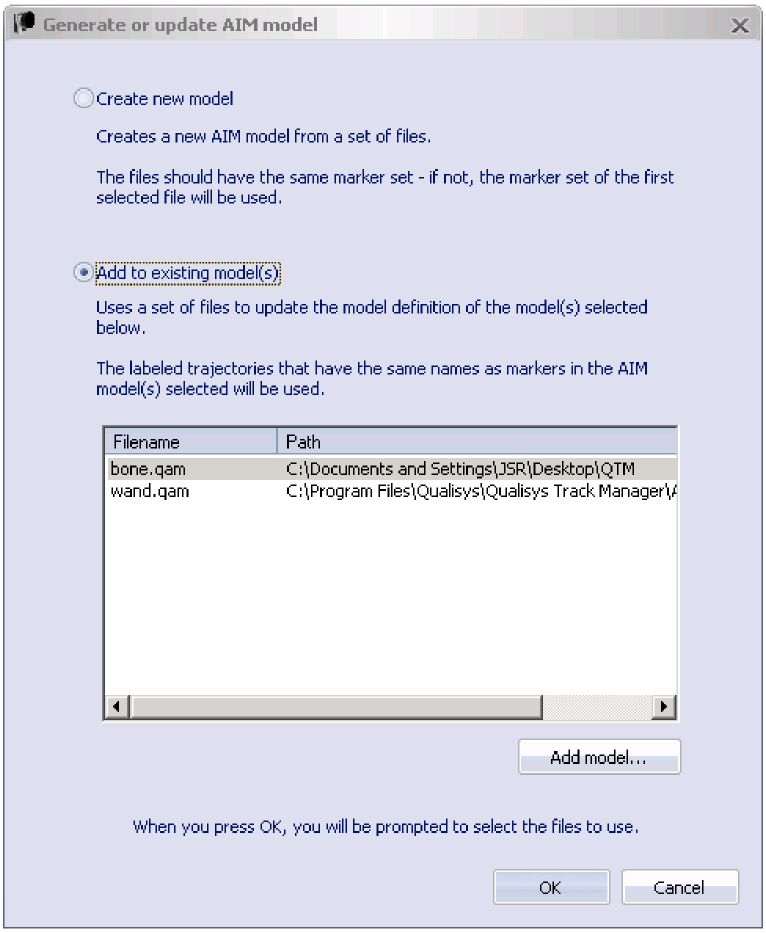
1. Make a measurement and identify all of the trajectories that will be included in the model. The model becomes better if you follow the guidelines in “Guidelines for data added to AIM models” on page 304: The two most important guidelines are:
    - Make sure that the subject in the file is moving. The movement does not have to be exactly the same as the rest of the captures, just that it includes most of the movement. When you add more measurements an AIM model, it becomes less important that the added files include a lot of movement.

- Make sure that the trajectories have the correct identity throughout the file. You can select a smaller measurement range to delete the incorrect parts if you do not want to identify the whole file.

2. Select the measurement range of frames that will be used in the model. Use the scroll boxes on the **Timeline control** bar to select the range. Choose a range where the motion is typical for what you are going to capture, most of the time it is best to use the whole measurement range unless there are large gaps or incorrect data.



3. Click the **Generate model** icon on the AIM toolbar or click **Generate model** on the **Auto ID** menu.
4. Select whether you want to **Create new model** or **Add to existing model**. Use add to existing model if you already have an AIM model with the current marker setup.



Create new model
Select the directory where you want to save the model, e.g. the same folder as the capture files. Then enter a **File name** for the model and click OK. The model is generated automatically.

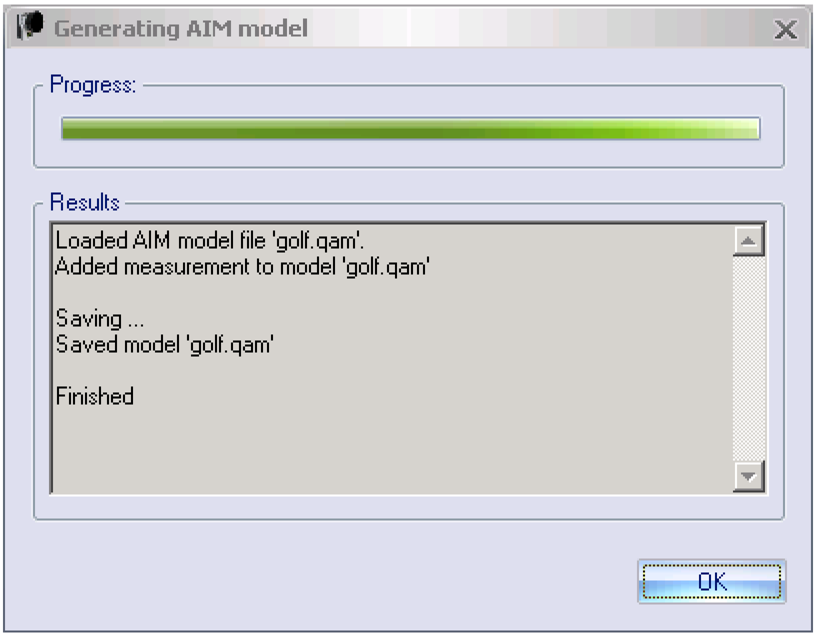
Add to existing model
Select a model from list to which you want to add the movement in the

current file. This option can be used to extend the motion of an existing model and therefore improve how it is applied to a new subject. It is the recommended option if you already have a model with the same marker setup. Make a copy of the original AIM model if you want to keep it.

The models in the list are the models that are available on the **AIM** page. By default all the models in the **Applied models** list are selected. Click Add model to browse for another model.

The file must include all the labeled trajectories that are included in the model. It can however have other labeled trajectories as well, only those with the same name as in the model will be used to update the model. This means that you can update several models at the same time. E.g. if the file include three person with different names on their respective labeled trajectories, then you can select all three AIM models in the list at once and all three models will be updated.

5. The dialog below is displayed when the AIM model has been created or updated. If anything goes wrong in the process it will be displayed in the **Results** list.



If you created a new AIM model it is then loaded to the **Applied models** list on the **AIM** page in the **Project options** dialog. Any other model in this list will be moved to **Previously used models** list.

If the data is added to an existing model, then the model files are updated and nothing is changed on the **Applied models** or **Previously used models** lists.

The model can now be applied to other measurements.

Note: The procedure is the same if you use the **Generate from multiple files** option on the **AIM** menu, except that you select the files you want to use in a dialog. However it is important that the files have the same labeled trajectories.

Guidelines for data added to AIM models

Follow these guidelines when you check the data before adding it to or creating an AIM model.

- Make sure that the subject in the file is moving. The movement does not have to be exactly the same as the rest of the captures, just that it includes most of the movement. When you add more measurements an AIM model, it becomes less important that the added files include a lot of movement.
- Make sure that the trajectories have the correct identity throughout the file. You can select a smaller measurement range to delete the incorrect parts if you do not want to identify the whole file.
- The colors of the trajectories and any bones between them are also saved in the model. For example to make it easier to verify the identification after AIM has been applied, the colors of the trajectories can be set with **Set different colors** on the **Trajectory info window** menu before creating the AIM model.
- Trajectories that are left unidentified or discarded will not be included in the model.
- If you have several subjects it is recommended to make an AIM model for each subject, see “AIM models for multiple subjects” on page 308.

The following two steps are not as important when you add files to the AIM model.

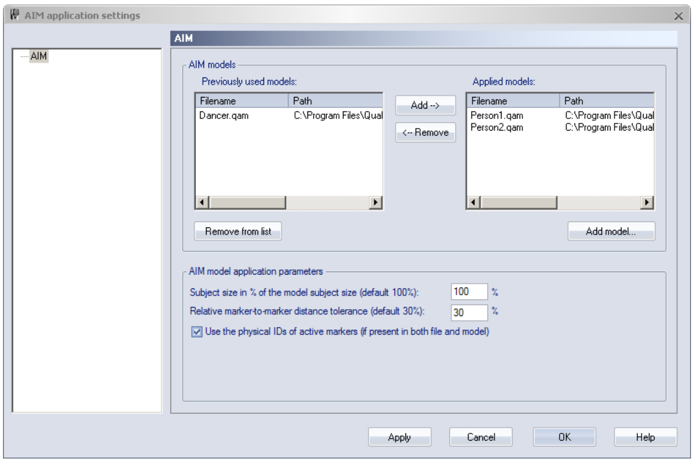
- The AIM model will be better if all of the trajectories are visible throughout the whole measurement. Therefore it is a good idea to gap-fill the trajectories as long as the gaps are relatively small. However if there are large gaps it is sometimes better to omit the gaps by limiting the measurement range in step 2 below. This is especially important if you use clusters, because then you have several trajectories that move along the same path.
- The data of each trajectory should be as good as possible. This is especially important if your model includes trajectories that are close to each other. Then a small erratic noise on a trajectory can make two trajectory paths come very close to each other, which makes the identification difficult. Therefore if the data is erratic, i.e. the trajectory movement is not smooth when you play the file, you should delete the erratic data by the following process:
    - Find the frame where the erratic data starts. Split the trajectory at that frame, see “Split part after current frame” on page 71.
    - Then step through the frames to locate where the trajectory data is OK again. Split the trajectory again.
    - Delete the part with erratic data that you have just created.
    - Repeat these steps for all of the frames where you can find erratic data and then gap-fill the trajectories

Applying an AIM model

The AIM model can be applied to files with captured motions that are similar to any part of the motion in the model. I.e. if your model includes a lot of different motions made by a human, then the captured trajectories can be identified if another human makes one or more of these motions. An AIM model can be applied either as a processing step or manually.

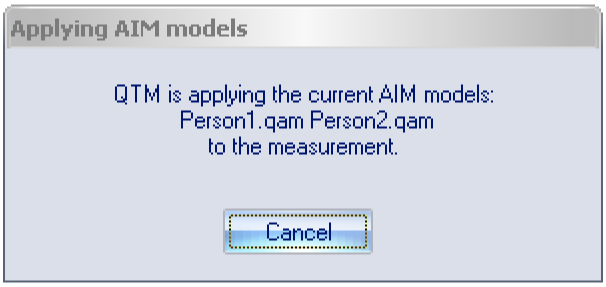
Below follows a description of how to apply an AIM model manually on a capture file.

1. Open a capture file in QTM.
2. Make sure that all of the trajectories are in the **Unidentified trajectories** window or the **Identified trajectories** window. Discarded trajectories are not used by AIM. It is also important that all required parts of the measurement is included in the selected measurement range, since the AIM model is only applied to trajectories with parts within the selected measurement range.
3. Click the **Apply model** icon on the **AIM** toolbar or click **Apply model** on the **AIM** menu. The **AIM application settings** dialog is displayed.

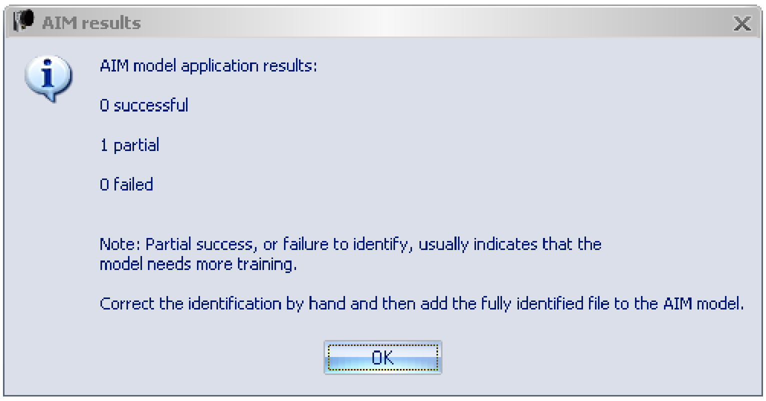


4. Check that the **Applied models** are correct. It is possible to apply several AIM models to the same file, for more information see “AIM models for multiple subjects” on page 308.

   You can also change the **AIM model application parameters**, see “AIM model application parameters” on page 178. However if you add measurements to the AIM model you will not have to change the parameters.
5. Click **OK**, the AIM module will try to apply the model to the measurement. Click **Cancel** in the dialog below to abort AIM.



   If any of the models cannot be applied to the trajectories the dialog below will appear. Showing how many of the bodies (models) were applied.



The **AIM results** dialog will display the result of all of the applied AIM models. The **Partial** results are AIM models where not all of the markers have been identified. For the **Failed** models none of the markers have been identified.

The most likely reason for the **Partial** result is that the model on the **AIM** page is incorrect for the captured motion. However, if the AIM model is correct it is recommended to manually identify the file and then add it to the existing AIM model, see “Generating an AIM model” on page 302.

It can also help to reduce the selected measurement range so that the AIM model is applied only on a smaller part of the measurement. For example, if the subject walks in and out of the volume it can help to reduce the selected measurement range to where the subject is inside the volume.

6. Check the file to see whether all of the trajectories have been identified correctly. There are two main reasons that trajectories are not identified.
   - AIM cannot match two trajectories if they have overlapping parts.
   - AIM sometimes cannot identify a trajectory if a nearby trajectory is missing during that part of the measurement.

   AIM does not correct any mistakes made by the tracker. If any trajectories switch identity in the measurement, this has to be corrected manually, preferably before applying an AIM model. These mistakes are not common, but are more likely to occur at the end of a trajectory. If you find such a mistake, split the trajectory and delete the parts that are wrong or move correct parts to a new trajectory. Then apply AIM again. Do not track the measurement again because then all of your editing will be removed.

7. When the AIM model has been successfully applied, the capture file must be saved to keep the changes.

When you apply a model as a processing step, either directly after a capture or in a batch process, it works exactly like when applying it manually. The model is set on the **AIM** page in the **Project options** dialog. The only difference is that there is no dialog displayed if AIM fails to find a solution to the identification problem.

How AIM identifies trajectories

AIM identifies trajectories from angles and distances between markers. In the AIM model QTM has saved the angle and distance ranges of added measurement files. This is the reason why it works better when you add more measurements since the AIM model will then include more movements. When the model is applied AIM tries to find the solution that fits these ranges the best.

There are a few limitations to AIM that are good to know:

1. AIM does not work on only two markers. The main reason is that AIM works in local coordinates, i.e. it does not take into account whether "marker x is to the left of marker y". AIM only cares about "what location marker x has compared to the locations of markers y and z". This means that there is no left/right, over/below or in front of/behind in the model.
2. AIM identifies the whole current measurement range at once and removes the identity of the previously identified trajectories. Therefore it is not possible to identify a measurement by applying AIM at different parts of the measurement.
3. AIM only looks for one solution. This means that AIM will only identify the trajectories once. If the measurement subject leaves the measurement volume completely and then comes back into it during the same measurement, only one of those two parts of the measurement can be identified by AIM.
4. AIM will not join two trajectories that overlap in time.
5. Because AIM is working with relations between markers, all of the markers in the model file must be included in the measurement to ensure that all of the trajectories can be identified. E.g. if a marker on the shoulder disappears for a long time it is difficult for AIM to find the arm.

Qualisys AB is continuously working on improving AIM so please contact us if you have any questions.

AIM models for multiple subjects

When capturing multiple subjects you must make a separate AIM model for each subject. Follow the process below to generate and apply the AIM models. For more information on how to generate and apply AIM models see “Generating an AIM model” on page 302 and “Applying an AIM model ” on page 305.

1. Make sure that there is marker pattern that differs between the different subjects . For example by placing four markers on the chest in different patterns.
2. Make sure that the label name of the subjects are different so that QTM can identify the labels when you add measurement to the AIM models.
3. The first time you create the AIM models you must identify one subject at a time and create a model. Then remove all of the labeled trajectories and repeat the process for the next subject and so on.
   Note: Empty labels are included in the AIM model so it is important to remove the whole label not just the data.

   However when you want to add more measurements to the AIM models, then you can add data to multiple AIM models at the same time. I.e. you can identify all of the subjects in the file and then select the AIM models you want to add the data to under the **Add to existing model(s)** option. The data will then be added to the AIM model with matching label names.
4. To apply multiple AIM models to a measurement you just add them to the **Applied models** list on the **AIM** page in the **Project options** dialog.
   Note: When you create a new AIM model all of the other models in the list are moved to the **Previously used models** list.

Events

Adding events

Events can be used to mark something that is happening. The events is sent in RT and can be added to a QTM file during the measurement and after the measurement.

There are two ways to create an event.

Trigger event (Oqus)
With the Oqus camera you can generate an event by pressing the trigger button during a measurement. It is recommended that you release the button quite quick, because releasing the trigger button can sometime also generate an event. If you have trouble with extra events, you can increase the hold-off time on the **Advanced (Timing)** page.

The event functionality is activated by default but can be changed with the **Generate event** setting for the external trigger on the **Advanced (Timing)** page, see chapter “External trigger (advanced)” on page 133.

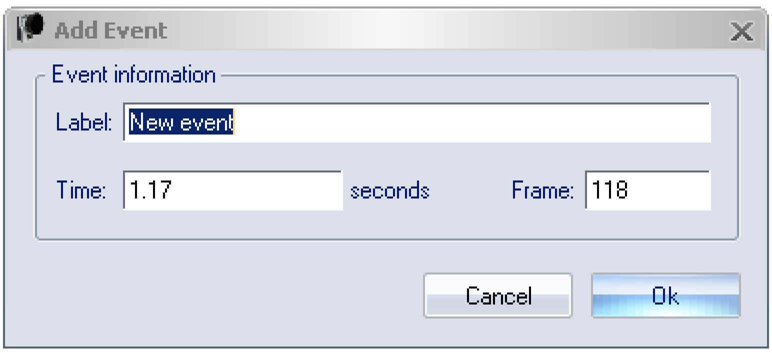
Note: Events cannot be created if you have selected to stop on the external trigger.

When the events are created during a capture it will be stored in the QTM file as **Trigger** event. The timing of the **Trigger** event will be the exact time when the signal is received by the camera. It is therefore the most exact way to set an event, especially if the trigger signal is generated automatically. Because the time of the event can be any time between two frames, it means that it is most likely not placed exactly at the capture of a frame. The frame number of the event will be rounded to the nearest frame.

Manual event
The manual event is created with the **Add event** button or with Ctrl+E. The events can be created with the button both during RT/capture and after a measurement.

When the events are created during a capture it will be stored in the QTM file as **Manual** event. The timing of the **Manual** event will be the time when you press the button in QTM. This means that it is most likely not placed exactly at the capture of a frame. The frame number of the event will be rounded to the nearest frame.

Events created with **Add event** button in a file will open the **Add event** dialog. The event will be placed on the current frame in the file. You can change the **Label** of the event and also **Time** and **Frame**.



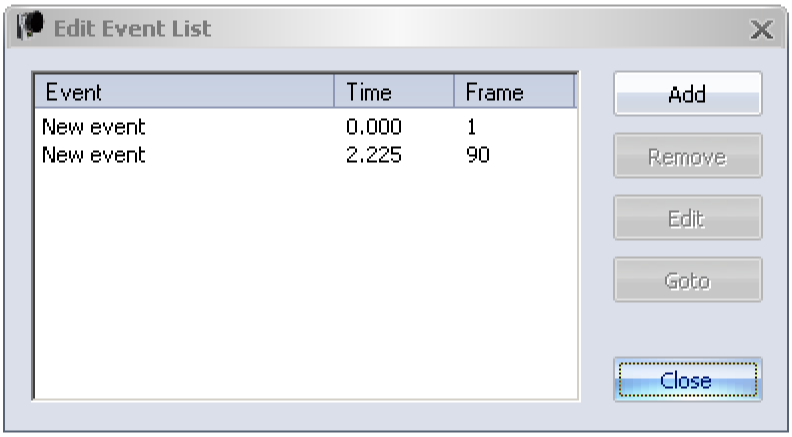
Viewing and editing events

The events are displayed above the timeline at the time of the event as a red triangle . In plots the event is displayed as a red line at the time of the event. Place the mouse over the event on the timeline to see information about the event. For information about all of the events open the **Edit event list** dialog, for example by right-clicking on an event and then on **Edit event list**.

The Label, Time and Frame of events in a file can be edited in the following ways.

Right-click on an event in the **Timeline control** bar and select **Edit event**. It will open the **Edit event** dialog where you can change the **Label**, **Time** and **Frame**.

Then you can access all of the events in the **Edit event list** dialog. Which can be opened by right-clicking on an event and then on **Edit event list**.



Add
Add a new event with the **Add event** dialog.

Remove
Remove the selected event.

Edit
Open the **Edit event** dialog where you can change the **Label**, **Time** and **Frame**.

Goto
Goto the frame of the event in the file.

Exporting events

The events can be exported to other programs with the C3D and TSV export. In the C3D export the events are always included. The format of the C3D file follows the C3D standard.

For the TSV export you must activate the **Include events** option to export the events, for information about the TSV format see chapter “Motion data (.tsv)” on page 315.

Force data calculation

Calculating force data

The force data is calculated from the analog signals of the force plate. It uses the parameters of the force plate on the **Force plate** page in the **Project options** dialog to calculate the correct values, see chapter “Force plate” on page 191. Force data can be calculated as a processing step or manually with the **Recalculate forces** icon F.

Note: It is important to not stand on the force plate at the start of the measurement, if you use the **Remove offset/drift** option on the analog data. It is also important to not stand on a Kistler force plate when the reset signal is sent. The reset signal is sent at the following operations in QTM:

New file

Changing a setting in Project option in QTM during preview

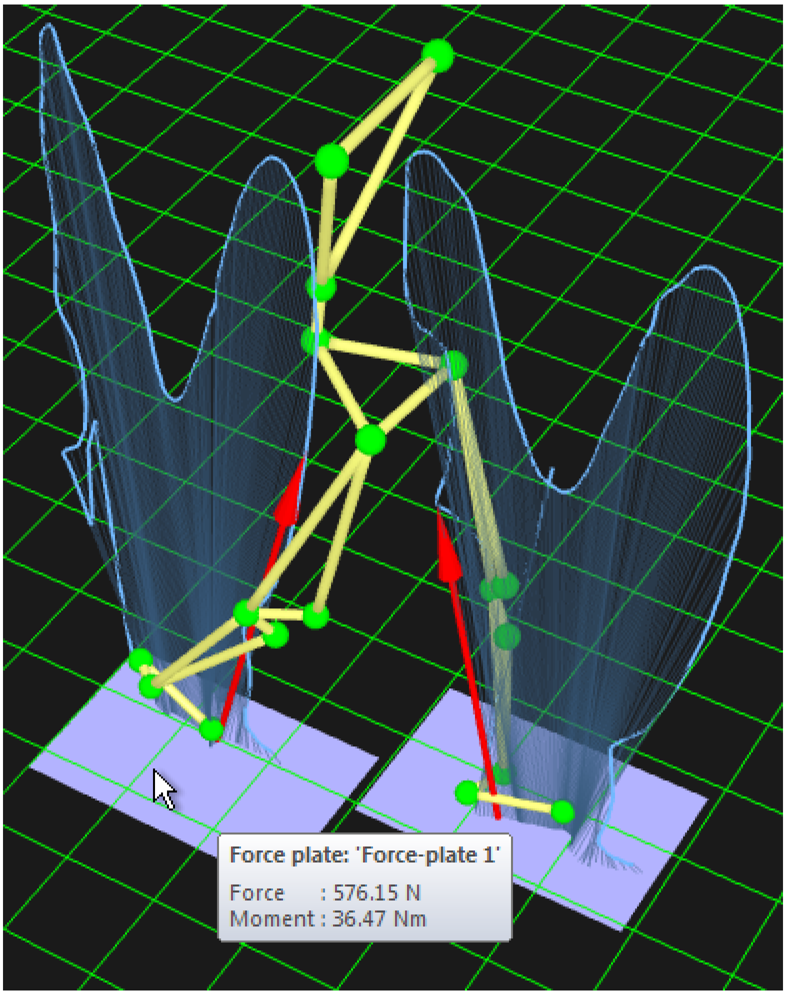
Just before the Start capture dialog is opened before a capture

In a batch capture just before QTM starts Waiting for next measurement/trigger

When the **Recalculate forces** command is used the settings of the force plate can be changed in the **File reprocessing** dialog. The settings in the dialog are the same as when the file was created. However, if the settings were not specified for the motion capture the current settings in the **Project options** dialog are copied instead.

Viewing force data

The force data in a capture file can be viewed in the **Data info** window and as a vector in the **3D view** window. The data can be used to confirm that the analog capture works and that the settings for the force plate are correct. For information about the data in the **Data info** window and how to plot it see chapter “Force data information” on page 83.



In the **3D view** window the force data is displayed as a vector with its base in the Center Of Pressure (COP). The vector displays the opposite force applied to the force plate, that is the reaction force. The purple squares represent the force plates and are placed at the force plate coordinates that are specified on the **Force plate** page in the **Project options** dialog. The light blue force traces (Pedotti diagrams) display the force data for the selected measurement range. Place the mouse over a force plate or force arrow to view a tool tip with the force data in the current frame.

The display of force arrows, plates and traces can be toggled on the **3D view settings** page in the **Project options** dialog. The color and size of the force can also be changed on this page.

Note: The force plates that are activated on the **Force data** page will be shown in the **3D view** window even if there is no analog data. So you can show a force-plate even if there is no analog data. Which can be used if the force is collected by another program, but you want to see the force plate location in QTM.

Note: If you transform the coordinate system the force plate coordinates will be the same, which means that you have to change them to move the force plate to the correct location, see chapter “Force plate location” on page 199.

Note: The Z direction of the force plate coordinate system is pointing down.

To make the most of the force data it can be exported to an analysis software. The best format to use are TSV, C3D or MAT file, because then the force plate location and for TSV and MAT the force data is included in the file, see chapter “Data export” on page

314. For example Visual3D uses C3D and recalculates the forces from the original data, therefore it can differ some from what is displayed in QTM.

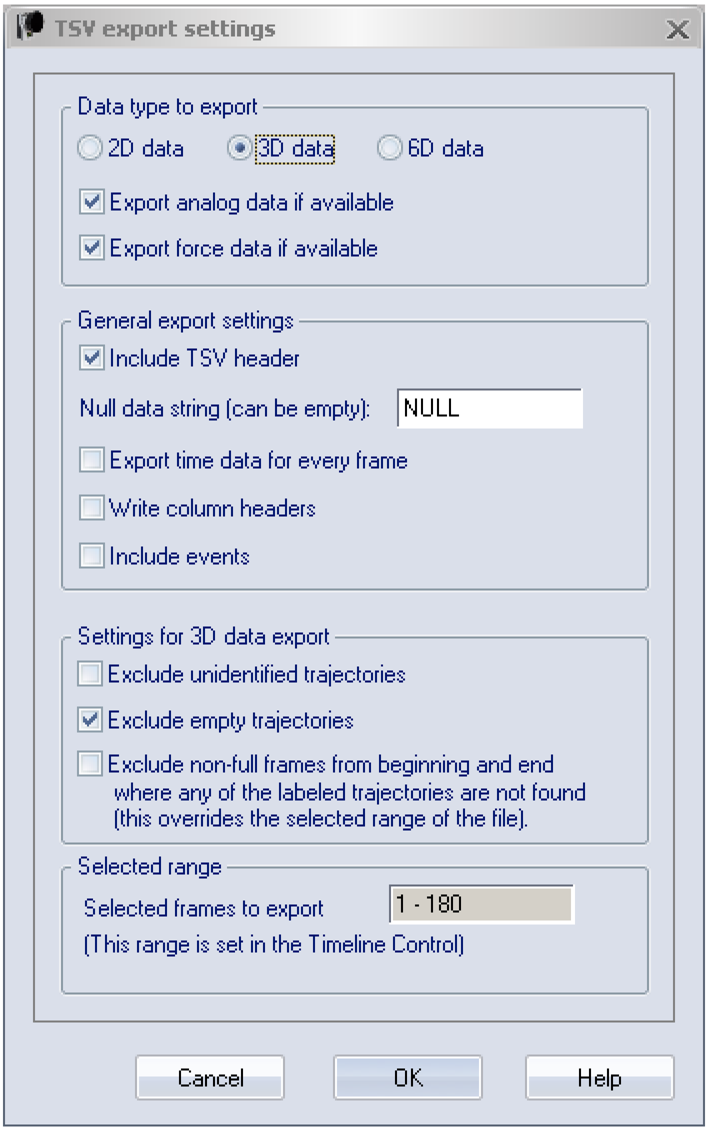
Data export to other applications

Data export

Exported measurement data can be analyzed in other applications. The data can be exported in three different ways: to TSV format, to C3D format and directly into Matlab. The data export can be done either as a processing step or manually. The file formats that will be the exported during processing are selected on the **Processing** page in the **Project options** dialog, see chapter "Processing" on page 170.

Export to TSV format

By exporting the data to TSV format you can analyze the data in any other program reads text files, e.g. Excel. Click **Export** on the **File** menu and then **To TSV** to export to the TSV format (Tab Separated Values).



For information about the settings see chapter "TSV export" on page 208. The frames that are included in the export are shown under the **Selected range** heading. The range is set by the measurement range on the **Timeline control** bar.

TSV file formats

There are four TSV file formats: 3D/2D (motion) data, 6DOF data, analog data and force data. The file formats are described in the chapters below.

The default TSV export includes the TSV header information and the 3D or 6DOF data in text format, depending on the capture. If analog and force data is included in the motion capture, another file which contains the analog data one for the force data will also be exported. 2D data can also be exported to a TSV file.

Note: A capture file called **filename.qtm** will be exported as **filename.tsv**, **filename_a.tsv** and **filename_f.tsv** (if the file contains analog data respectively force data). The file name can however be changed during the export.

Motion data (.tsv)

The motion data file contains the data of the trajectories of the motion capture and it has two parts: file header and data. It can contain either 2D or 3D data depending on the settings on the **TSV export** page in the **Project options** dialog.

In the file header, the variable names are written as one word (without any embedded spaces) followed by a tab character and the variable value as a second word. Each variable is on a new line. The following variables are available in the file header:

NO_OF_FRAMES
Total number of frames in the exported file.

NO_OF_CAMERAS
Number of cameras used in the motion capture.

NO_OF_MARKERS
Total number of trajectories (markers) in the exported file.
Note: Only included in 3D data export.

FREQUENCY
Measurement frequency used in the motion capture.
Note: When using external timebase the frequency is set to the actual frequency for the **Multiplier/Divisor** mode and to EXT for the **Edge triggered** mode.

NO_OF_ANALOG
Not in use by QTM.
Note: Only included in 3D data export.

ANALOG_FREQUENCY
Not in use by QTM.
Note: Only included in 3D data export.

DESCRIPTION
At present not in use by QTM.
Note: Only included in 3D data export.

TIME_STAMP
Date and time when the motion capture was made. The date and time is followed by a tab character and then the timestamp in seconds from when the computer was started.

DATA_INCLUDED
Type of data included in the file, i.e. 2D or 3D.

EVENT
Each event is added on a new row starting with word EVENT. Then followed by the name of the event, frame number and time, each separated by a tab character.
Note: The events are only added if **Include events** are active for the TSV export settings.

MARKER_NAMES
List of trajectory labels (trajectories in the **Labeled trajectories** window) separated by tab characters. Unidentified trajectories have no names and are therefore only represented by a tab character. The number of names in the list corresponds to the value given by the **NO_OF_MARKERS** variable.
Note: Only included in 3D data export.

In 3D export the data part follows on a new line after the last marker name. The trajectory data (in mm) is then stored in tab-separated columns, where every row represents one frame. Each trajectory has one column for each direction (X, Y and Z). The data for the first marker is therefore stored in the first three columns and data for the second marker is stored in column 4, 5 and 6 etc.

In 2D export the data part follows on a new line after the variable **DATA_INCLUDED**. The data part starts with a row with the number of the cameras. The marker data for each camera is then given below its corresponding heading, e.g. **Camera: 1**. Every marker has four columns with data, which is the same as that shown in the **Data info** window: **x**, **y**, **xSize** and **ySize**. Each row in the marker data represents one frame and it is just the markers that are visible in that frame that are included. The order of the markers can therefore not be used to identify the markers.
Note: The 2D data in the TSV export is linearized in contrast to that in the **Data info** window. Therefore the 2D data will differ between the two.

There are two options in the TSV export with which you can add more information to the file. First the option **Export time data for every frame** will add the time data to the first two columns. The first column contains the frame number and the second contains the time. Then with the **Write column header** option you can add a header above each column describing the contents of that column.

Analog data (_a.tsv)

The analog data files contain the data of the analog capture. Each file contains the data from one analog board or EMG system. If there is only one source of analog data the file name ends with _a. If there are more than one source of analog data the files are numbered in the same order as they appear in the **Data info** window (_a_1, _a_2 and so on). The file is exported in 2D, 3D and 6DOF export, if the **Export analog data if available** option is activated.

There are two parts in the file: file header and data. In the file header, the variable names are written as one word (without any embedded spaces) followed by a tab character and the variable. Each variable is on a new line. The following variables are available in the file header:

NO_OF_SAMPLES
Total number of samples in the exported file.

TOT_NO_OF_CHAN
Total number of channels in the exported file.

FREQUENCY
Sampling frequency of the analog measurement.

NO_OF_CALC_CHAN
At present not used by QTM.

TIME_STAMP
Date and time when the measurement was made. The date and time is followed by a tab character and then the timestamp in seconds from when the computer was started.

FIRST_SAMPLE
Original number of the first frame in the range that is exported from the QTM software. The start time of the exported data can then be calculated as **FIRST_ SAMPLE** / **FREQUENCY**.

DESCRIPTION
At present not used by QTM.

DATA_INCLUDED
Type of data included in the file. Set to **ANALOG** by the QTM software

CHANNEL_NAMES
List of tab-separated channel names. The number of names in the list corresponds to the number of channels given by the **TOT_NO_OF_CHAN** variable.

CHANNEL_GAIN
List of tab-separated gains of each channel.
At present they are all set to 1 by QTM.

FP_LOCATION, **FP_CAL**, **FP_GAIN**
At present not used by QTM.

The data part follows on a new line after **FP_GAIN**. The data of the analog channels are then stored in tab-separated columns, one column for each channel and one row per sample. The data is always saved with 6 digits.

There are two options in the TSV export with which you can add more information to the file. First the option **Export time data for every frame** will add the time data to the first two columns. The first column contains the frame number and the second contains the time. Then with the **Write column header** option you can add a header above each column describing the contents of that column.

Force data (_f.tsv)

The force data files contain the data of the force plates. Each file contains the data from one force plate. If there is only one force plate the file name ends with _f. If there are more than force plate the files are numbered in the same order as they appear on the **Force data** page (_f_1, _f_2 and so on). The file is exported in 2D, 3D and 6DOF export, if the **Export force data if available** option is activated.

There are two parts in the file: file header and data. In the file header, the variable names are written as one word (without any embedded spaces) followed by a tab character and the variable. Each variable is on a new line. The following variables are available in the file header:

NO_OF_SAMPLES
Total number of samples in the exported file.

FREQUENCY
Sampling frequency of the force data.

TIME_STAMP
Date and time when the measurement was made. The date and time is followed by

a tab character and then the timestamp in seconds from when the computer was started.

FIRST_SAMPLE
Original number of the first frame in the range that is exported from the QTM software. The start time of the exported data can then be calculated as **FIRST_SAMPLE / FREQUENCY**.

DESCRIPTION
At present not used by QTM.

DATA_INCLUDED
Type of data included in the file. Set to **Force** by the QTM software

FORCE_PLATE_TYPE
The type of force plate.

FORCE_PLATE_NAME
The name of the force plate defined on the **Force data** page.

FORCE_PLATE_CORNER_POSX_POSY_X, **FORCE_PLATE_CORNER_POSX_POSY_Y** and **FORCE_PLATE_CORNER_POSX_POSY_Z**
Position of the top left corner when looking at the internal force plate coordinate system. The position is in the measurement coordinate system.

FORCE_PLATE_CORNER_NEGX_POSY_X, **FORCE_PLATE_CORNER_NEGX_POSY_Y** and **FORCE_PLATE_CORNER_NEGX_POSY_Z**
Position of the top right corner when looking at the internal force plate coordinate system. The position is in the measurement coordinate system.

FORCE_PLATE_CORNER_NEGX_NEGY_X, **FORCE_PLATE_CORNER_NEGX_NEGY_Y** and **FORCE_PLATE_CORNER_NEGX_NEGY_Z**
Position of the bottom left corner when looking at the internal force plate coordinate system. The position is in the measurement coordinate system.

FORCE_PLATE_CORNER_POSX_NEGY_X, **FORCE_PLATE_CORNER_POSX_NEGY_Y** and **FORCE_PLATE_CORNER_POSX_NEGY_Z**
Position of the bottom right corner when looking at the internal force plate coordinate system. The position is in the measurement coordinate system.

FORCE_PLATER_LENGTH
The length of the force plate.

FORCE_PLATE_WIDTH
The width of the force plate.

The data part follows on a new line after **FORCE_PLATE_WIDTH**. A header is always included above each column describing the contents of that column. The data of the force plate are then stored in tab-separated columns, in the following order Force_X, Force_Y, Force_Z, Moment_X, Moment_Y, Moment_Z, COP_X, COP_Y, and COP_Z and one row per sample. The data is in the internal coordinate system of the force plate and always saved with 6 digits.

There are two options in the TSV export with which you can add more information to the file. First the option **Export time data for every frame** will add the time data to the first two columns. The first column contains the frame number and the second contains the time.

6DOF data format

The TSV export of files with 6DOF data creates a TSV file (.tsv) with a file header and a data part. The variable names in the file header are followed by a tab character and then the value. Each variable is on a new line.

NO_OF_FRAMES
Total number of frames in the exported file.

NO_OF_CAMERAS
Number of cameras used in the motion capture.

NO_OF_BODIES
Total number of rigid bodies in the exported file.

FREQUENCY
Measurement frequency used in the motion capture.
Note: When using external timebase the frequency is set to the actual frequency for the **Multiplier/Divisor** mode and to EXT for the **Edge triggered** mode.

NO_OF_ANALOG
Not in use by QTM.

ANALOG_FREQUENCY
Not in use by QTM.

DESCRIPTION
At present not in use by QTM.

TIME_STAMP
Date and time when the motion capture was made. The date and time is followed by a tab character and then the timestamp in seconds from when the computer was started.

DATA_INCLUDED
Type of data included in the file, i.e. 6D.

EVENT
Each event is added on a new row starting with word EVENT. Then followed by the name of the event, frame number and time, each separated by a tab character.
Note: The events are only added if **Include events** are active for the TSV export settings.

BODY_NAMES
Tab-separated list with the names of the rigid bodies in the exported file.

On a new line after the last rigid body name follows a tab-separated list of the data headings for the rigid bodies. The headings are:

X, Y and Z
The position of the origin of the local coordinate system of the rigid body. Where X, Y and Z are the distance in mm to the origin of the coordinate system for rigid body data, see chapter “Coordinate system for rigid body data” on page 184.

Roll, Pitch and Yaw
Roll, pitch and yaw of the rigid body in degrees.
Note: The names and their definition will change if the definition is changed on the **Euler angles** page in the **Project options** menu.

Residual
The average of the errors (in mm) of each measured marker compared to the 6DOF body definition. This error is probably larger than the 3D residual.

Rot[0] - Rot[8]
The elements of the rotation matrix for the rigid body. Where the elements are placed in the matrix according to the following table:

Rot[0]	Rot[3]	Rot[6]
Rot[1]	Rot[4]	Rot[7]
Rot[2]	Rot[5]	Rot[8]

Note: For information about the rotation matrix see “Rotation angle calculations in QTM” on page G - 1.

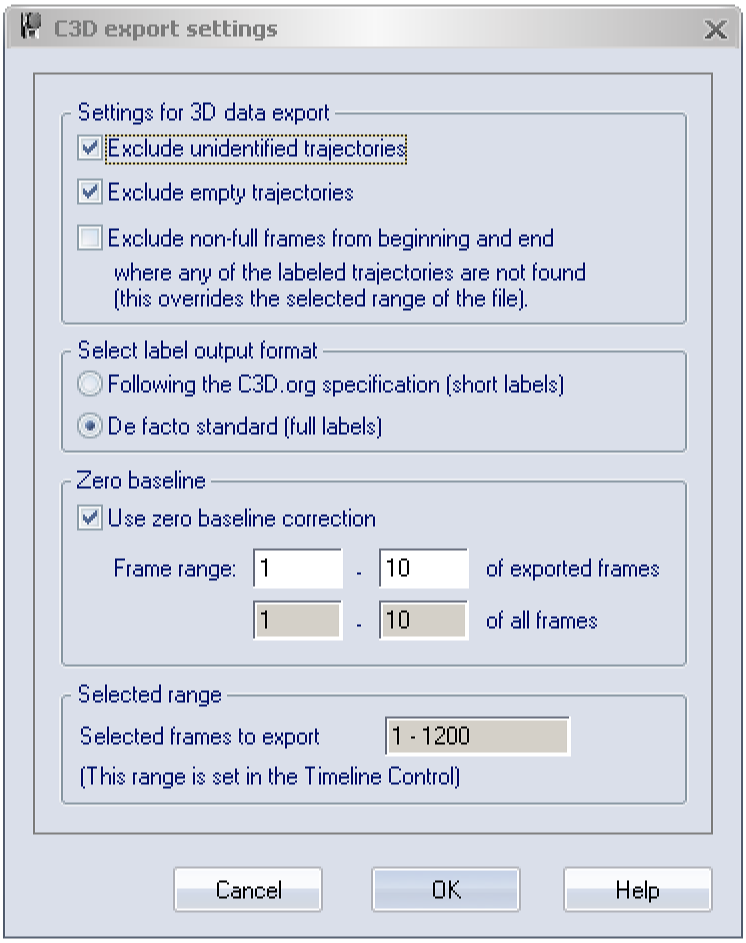
The data part follows on a new line after **Rot[8]**. The data is stored in tab-separated columns, where each row represents a frame. The columns are in the same order as the heading list described above. If there is more than one rigid body, their frames are stored on the same rows as the first body. They are just separated by two tab characters after the **Rot[8]** data of the previous body.

There are two options in the TSV export with which you can add more information to the file. First the option **Export time data for every frame** will add the time data to the first two columns. The first column contains the frame number and the second contains the time. Then with the **Write column header** option you can add a header above each column describing the contents of that column.
Note: Each rigid body name is only entered before its respective X column header, the following headers only includes the contents of the column.

Export to C3D format

Click **Export** on the **File** menu and then **To C3D** to export to the C3D format. The C3D format is a motion capture industry file format. It supports simultaneous storage of motion data, analog data and other data types in the same file. For further information about the file format see chapter “C3D file format” on the facing page.

Note: C3D data can be exported even if the file does not contain any 3D data. For example if you have measured forces or EMG.



For information about the settings see chapter “C3D export” on page 210. The frames that are included in the export are shown under the **Selected range** heading. The range is set by the measurement range on the **Timeline control** bar.

The frames used for **Zero baseline correction** is set in relation to the exported range. However the range is also displayed in relation to **all frames**. Make sure that the force plates are unloaded during the frames that are used for correction. Otherwise the data will be wrong in for example Visual3D.

C3D file format

The C3D export creates a C3D file, for information about the binary C3D format see http://www.c3d.org.

One limit of the C3D format is that the analog frequency must be the same for all analog data and a multiple of the marker frequency. If this is not the case then the analog data will be resampled at the multiple equal to or higher than the highest analog frequency in the measurement. For example if the marker frequency is 120 Hz, the EMG frequency is 1500 Hz and the analog frequency 1200 Hz. Then all of analog data will be resampled to 1560 Hz.

Note: A file called **filename.qtm** in QTM will be called **filename.c3d** after the export. The file name can however be changed during the export.

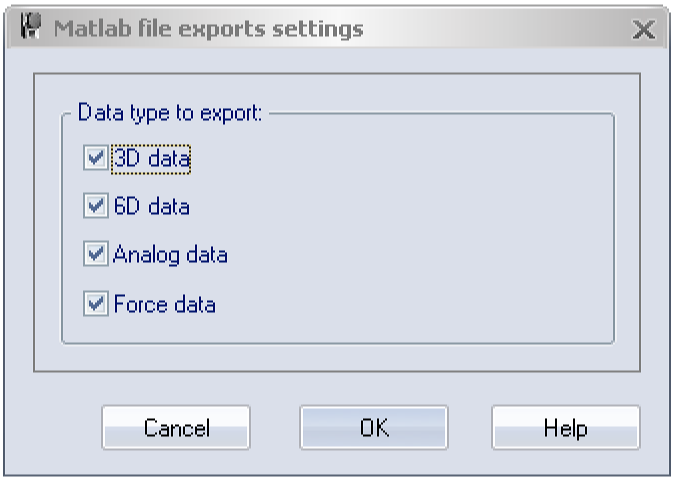
Export to DIFF format

The DIFF format is mainly used in Japan for transferring data for Gait Analysis, for the definition refer to the DIFF Data Interface File Format (DIFF) User's Manual. Follow these steps to export to DIFF.

1. Open a file with the wanted marker setup and keep it open. You can also apply an AIM model in RT to get the labels in the **Labeled trajectories** list.
2. Open the **Project options** dialog. For information about the DIFF export settings see chapter “DIFF export” on page 213.
3. Specify the labels that you have used on the **G1R**, **G1L**, **G2R** and **G2L** page. Any unused labels will be empty in the DIFF export. The label specifications are saved in the QTM workspace, so you only have to change these settings if your marker labels are changed.
4. Go to the **DIFF export** page and enter the information for the subject. These settings have to be entered for every new subject.
5. Capture the data of the subject.
   a. a.Use batch capture to enumerate the files.
   b. b.Make sure that the name of the file is the same as the Data id for the DIFF export, because the name of the exported DIFF file will be the same as the QTM filename.
6. Identify the markers, either by using AIM or manually.
7. Click **Export** on the **File** menu and then **To DIFF** to export to the DIFF format. You can review the settings before the export.
   Note. You can also export the DIFF data as a processing step, see chapter “Introduction to data processing” on page 278.

Export to MAT format

Click **Export** on the **File** menu and then **To MAT** to export to the MAT format. The MAT file can then be opened in Matlab. You can select the data that is exported on the **Matlab file export** page in the **Project options** dialog, see chapter “Matlab file export” on page 212. For further information about the file format see chapter “MAT file format” on the facing page.



Note: Because of Matlab limitations it is important that the file does not start with a number or a none English letter.

Note: The names of trajectories and analog channels must be longer than 3 letters. Shorter names will be extended with underscore so that they are 3 letters.

MAT file format

When the data from QTM is exported to a MAT file a struct array is saved in the file. The struct array is named the same as the file. Therefore it is important that the file does not start with a number or a none English letter.

The struct array contains four to seven fields depending on if the file includes analog, EMG and 6DOF data. The fields are **File**, **Frames**, **FrameRate**, **Trajectories**, **Analog**, **EMG**, **Force**and **RigidBodies**, that have the following contents:

File
File name and directory path.

Frames
Number of frames.

FrameRate
Frame rate in frames per second.
Note: When using external timebase the frequency is set to the actual frequency for the **Multiplier/Divisor** mode and to EXT for the **Edge triggered** mode.

Trajectories
Struct array with fields for the three **Trajectory info** windows: **Labeled**, **Unidentified** and **Discarded**. These fields are struct arrays with three fields:

Count
Number of trajectories in the window.

Labels
A list of the trajectory labels.
Note: This field is only included in the **Labeled** struct array.

Data
The location of the 3D points (in mm) of the trajectories in the window. The data is given in a matrix with the dimensions: Trajectories * X, Y, Z direction and Residual * Frames. The trajectories are in the same order as in the **Trajectory info** window.

Analog/EMG
Struct array with data from the analog capture. The analog and EMG data from integrated wireless EMGs are stored in separate struct arrays, but the data that is included is the same.
Note: This struct array is only included if the capture file has analog data.

BoardName
The name of the board that was used in the capture.

NrOfChannels
The number of channels that were used in the capture.

ChannelNumbers
The channel numbers that were used in the capture.

Labels
An array with the names of the channels that were used on the analog board.

Range
The range of the channels on the analog board.

NrOfFrames
The number of frames of the analog capture.

SamplingFactor
The multiplication factor compared with the motion capture frame rate.

NrOfSamples
The number of samples in the analog capture.

Frequency
The frequency of the analog capture.

Data
The data (in V) of the analog capture. The data is given in a matrix with the dimensions: Analog channels * Frames of the analog capture.

Force
Struct array with data from the force plates.
Note: This struct array is only included if the capture file has force data.

ForcePlateName
The name of the force plate that was used in the capture.

NrOfFrames
The number of channels that were used in the capture.

SamplingFactor
The multiplication factor compared with the motion capture frame rate.

NrOfSamples
The number of samples in the analog capture.

Frequency
The frequency of the analog capture.

Force
The force data in newton (N), the data is given for X, Y and Z direction.

Moment
The moment data in newton metre (Nm), the data is given for X, Y and Z direction.

COP
The centre of pressure on the force plate (in mm), the data is given X, Y and Z direction. The position is given in the internal coordinate system of the force plate

ForcePlateLocation
The location of the force plate in measurement coordinate system. The corners are in the order upper left, upper right, lower right and lower left seen in the force plate coordinate system.

RigidBodies
Struct array with data for the 6DOF bodies.
Note: This struct array is only included if the capture file has 6DOF bodies.

Bodies
The number of 6DOF bodies.

Name
The names of the 6DOF bodies.

Positions
The position of the origin of the measured rigid body's local coordinate system. It is given as a matrix with the dimensions: Bodies * Distances (X, Y and Z) * Frames. The distances are in mm to the origin of the coordinate system of the motion capture.

Rotations
The rotation matrixes of the rigid bodies. It is given as a matrix with the dimensions: Bodies * Rotation matrixes (elements 0-8) * Frames. The elements are placed in the matrix according to the following table:

[0]	[3]	[6]
[1]	[4]	[7]
[2]	[5]	[8]

Note: For information about the rotation matrix see “Rotation angle calculations in QTM” on page G - 1.

RPYs
The roll, pitch and yaw of each rigid body. It is given as a matrix with the dimensions: Bodies * Rotation angles (roll, pitch and yaw) * Frames. The rotation angles are in degrees.

Note: The matrix will always be called **RPYs** even if the definitions are changed on the **Euler angles** page in the **Project options** menu.

Residuals
The residual of the rigid bodies.

SMPTETimecode
Struct array with a list of all the SMPTE timecodes of the frames in the file.

Note: This struct array is only included if the capture file was captured with a SMPTE timecode.

Hour
The hour of the timecode.

Minute
The minute of the timecode.

Second
The second of the timecode.

Frame
The SMPTE frame counted from the start of the SMPTE synchronization source.

Subframe
Because the SMPTE timecode is at 30 Hz there can be more than one marker frame per SMTPE frame.

Missing
Indicates if the SMPTE timecode is extrapolated if the SMPTE synchronization source is lost during the measurement.

Events
Struct array with a list of all the events in the file.

Note: This struct array is only included if the capture file has events.

Label
The label of the event.

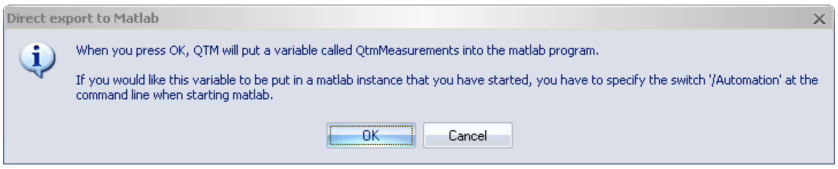
Frame
The corresponding frame of the event. The time will be rounded of to the nearest frame.

Time
The time of the event.

Note: To use the struct array, write the name of the struct array and the fields with a period between them. If several files have been exported to Matlab, write the variable as **QTMmeasurements(1)**, **QTMmeasurements(2)** and so on to get the data of the file.

Export directly into Matlab

The export directly into Matlab requires that Matlab is installed, since it is exported into a struct array in a Matlab command window. Click **Export** on the **File** menu and then **Directly into Matlab** to export to Matlab. The frames that are included in the export are set by the measurement range on the **Timeline control** bar.



The export creates a struct array called **QTMmeasurements** in the Matlab workspace, this includes all of the data that can be saved in the MAT file export, for information about the format see chapter “MAT file format” on page 323.

Data from several capture files can be exported to the same Matlab window, the data will just be added to the struct array. Use the Matlab command 'save' so that you can then open the struct array in a Matlab window with the regular GUI.

Euler angles

Defining Euler angles in QTM

Euler angles (rotation angles) are the way that QTM shows the rotation of a 6DOF body. It is also how you enter any rotation that should be applied to a global or a local coordinate system. It is therefore important to understand how Euler angles work to be able to use 6DOF data correctly.

Euler angles are a method to define the rotation of a body. The problem is that they are not unambiguous and therefore QTM internally uses a rotation matrix to define the rotations. This means that changing the Euler angles definition only changes the interpretation of the matrix. The rotation angles are transformed into the rotation matrix with the calculations described in chapter “Calculation of rotation angles from the rotation matrix (Qualisys standard)” on page G - 1. The same calculations are then used to acquire the measured rotation angles, since they can be more directly interpreted and visualized by the user.

Important: When you change the Euler angles definition that change will be effective immediately everywhere in QTM. This means for example that if you open a file after you have changed the definitions it will be displayed using the new definitions and not those used when the file was saved. It also means that the angles on the **Transformation** page will change to reflect the rotation of the global coordinate system with the new definition.

In QTM you can define the Euler angles as any possible rotation of a right-hand coordinate system, see chapter “Euler angles” on page 186. By default QTM uses the **Qualisys standard** definition, which is described in the chapter “Rotation angles in QTM” on page 292.

System hardware

Cameras

The camera types that can be used with QTM are Oqus, ProReflex MCU and MacReflex.

When the Qualisys ProReflex camera are used the firmware version must be 7.03.001 or higher. For detailed information about the ProReflex MCU see “ProReflex MCU - Camera manual” on page B - 1.

For information about MacReflex cameras check the Position server manual that was distributed with the camera system.

Oqus

The Oqus cameras consist of five series of cameras: 5-series, 4-series, 3+-series, 3-series and 1-series. For more information about the different series of the Oqus camera see “Oqus - Camera manual” on page A - 1.

The following chapters describe how to setup the Oqus camera system and other hardware information that is common for all of the Oqus-series.

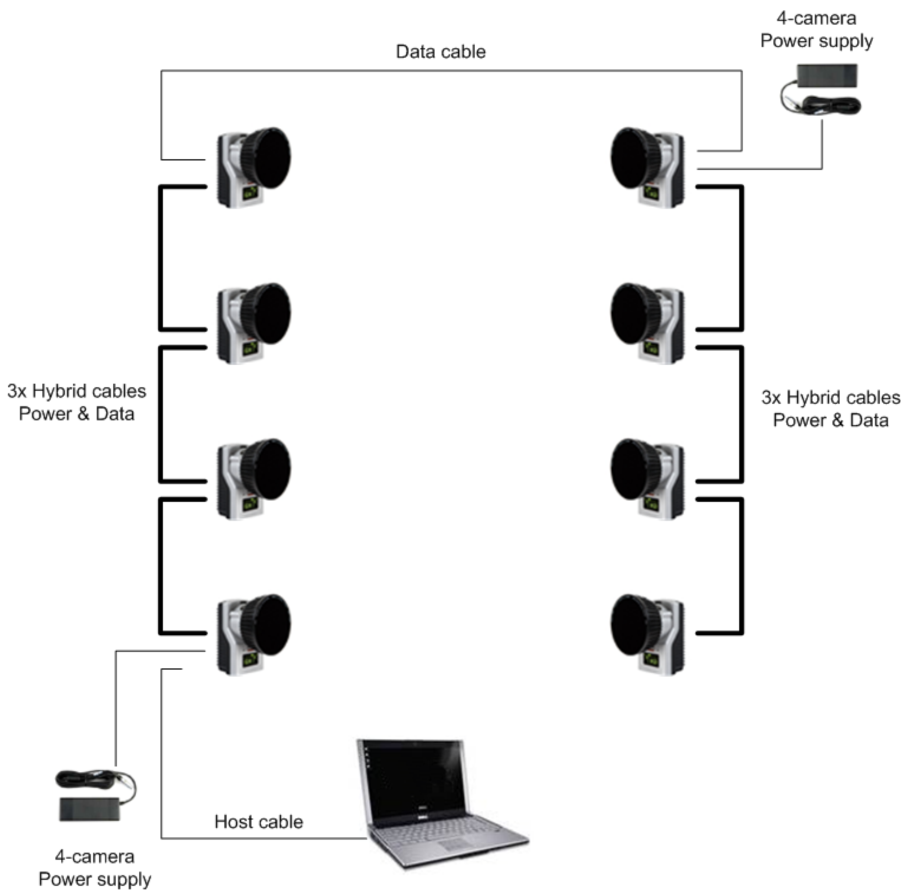
Setting up the system (Oqus)

Before you connect the Oqus camera system, make sure that the QDS (Qualisys DHCP server) is running and that the network interface settings are correct, see “QDS” on page 332 respectively “Network card setup” on page 341.

The Oqus system is easy to setup. The connectors are unique and cannot be connected to the wrong ports. Further, the connector color matches that of the port. The **DATA** connector can be connected to any of the two DATA ports, and the **POWER** connector can be connected to any of the two **POWER** ports, so it does not matter on which side you put the connector. For more information on the connectors, see “Oqus connectors” on next page.

Note: When the cables have been connected correctly the LEDs on the back of the Oqus will be lit. The **EXT** LED will be lit green and the **ACT** LEDs will be blinking.

However, the connection of the power adaptors do require some attention. One power adapter can power up to 4 Oqus cameras. Therefore, the connection of a camera system comprising of more than four cameras must look something like the image below.



This means that you must use the following cables for an 8 camera system:

1 host cable between computer and camera.

6 bundled cables with power and data.

1 data cable connected between camera 4 and 5 in the setup.

2 AC/DC adapters, connected to for example camera 1 and 5.

When the camera system has been connected continue with setting aperture and focus and then arrangement of the cameras, see “Setting the aperture and focus” on page 343 respectively “Arranging the cameras” on page 348.

Note: For Oqus systems larger than 16 cameras and for systems with many high-speed cameras, the performance can sometimes be improved with a gigabit switch and then connect the cameras in shorter daisy-chains, see “Setup Oqus system with Ethernet switch” on page 342.

Oqus connectors

The back of the camera holds six connectors for power, data and control connections. The view differs slightly depending on the type of camera. The image below shows the standard version of the camera. The water protected version uses different connectors and lacks the LEDs found on the standard version.

1. **Left Data port** (light blue)
   Ethernet connector. 100BaseTX/802.3i, 100 Mbps, Fast Ethernet.
2. **Right Data port** (light blue)
   Identical to the left data port
3. **Left Ethernet activity indicator**
   Shows the status of the Ethernet connection. Fixed green light means that a carrier signal has been detected and that the connection is up. Flashing green light indicates that data is received and/or transmitted.
4. **Right Ethernet activity indicator**
   Identical to the left indicator.
5. **Battery port** (white)
   Used to supply the camera with power from an Oqus compatible battery.
6. **Battery status indicator**
   Lit green when the camera is supplied though the **BATTERY** port.
   Lit red when a voltage outside the specified range (10-16V) is connected to the port.
7. **Power supply status**
   Lit green when the camera is powered through one of the **POWER** ports. A red light indicates internal power supply error.
8. **Right power supply port** (black)
   Daisy-chain power port. Supplies the camera with 48VDC and can be daisy chained to supply cameras further down the chain with power.
9. **Left power supply port** (black)
   Identical to the right power supply connector.
10. **Control port** (light grey)
    The control port is used to synchronize the camera with external sources, and contains pins for among other things external trigger in, external sync in and external sync out. Splitter cables are needed to connect one or more BNC

cables to this port, for more information see “Control connections” on page A - 9.

11. **Camera identification**
This label provides information on:

- The serial number of the camera
- The product number
- The Ethernet Mac address
- The WLAN Mac address

QDS

QTM comes with a DHCP-server called QDS (Qualisys DHCP Server), which distributes IP addresses to the Oqus cameras. An IP address for each camera is required to be able to communicate with them over the Ethernet network. QDS will be installed automatically with QTM and it must be running at all times, to provide the cameras with IP addresses at startup. The DHCP server will only give IP addresses to Oqus cameras so it will not disturb your computer network.

QDS includes a wizard to configure network interfaces to use them with Oqus cameras and to activate wireless operation on an Oqus camera, see “Oqus network configuration wizard” on the facing page and “Oqus wireless camera setup wizard” on page 335.

QDS menu

To open the QDS menu, right-click on the QDS icon in the status toolbar.

The QDS menu contains the following options:

Oqus configuration wizard
The Oqus configuration wizard can be used to setup your network interface for Oqus and to configure an Oqus camera for wireless operation, see “Oqus network configuration wizard” on the facing page and “Oqus wireless camera setup wizard” on page 335.

Advanced
Advanced configuration of network interfaces on the computer, see “Advanced (network settings)” on page 339.

Camera control utilities
QDS can control the Oqus cameras with the following commands.

Reboot all Oqus cameras
Reboot all Oqus cameras that have an IP-address.

Show IP address on camera display
Switch the display to show the IP-address and Serial number of the camera

Show ID on camera display
Switch the display to show QTM ID of the camera. This is the default for cameras that have been connected to QTM.

Network configurations
Using this sub-menu you can save or load network configurations. Click on **Save** to save the current configuration and then to load a configuration click on **Load**.

About QDS
Information about QDS.

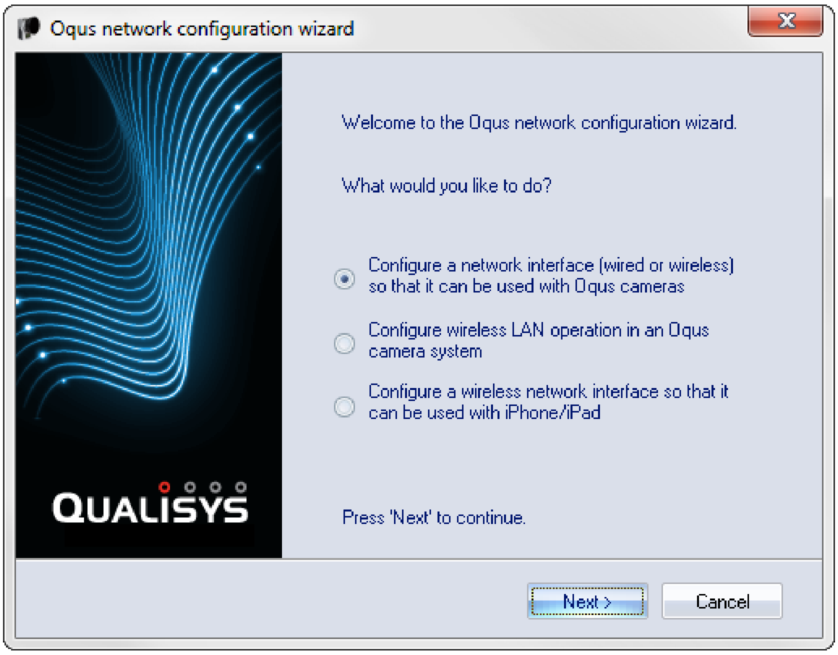
Start QDS automatically
Option for starting QDS automatically on computer startup.

Shut down
Shut down QDS.

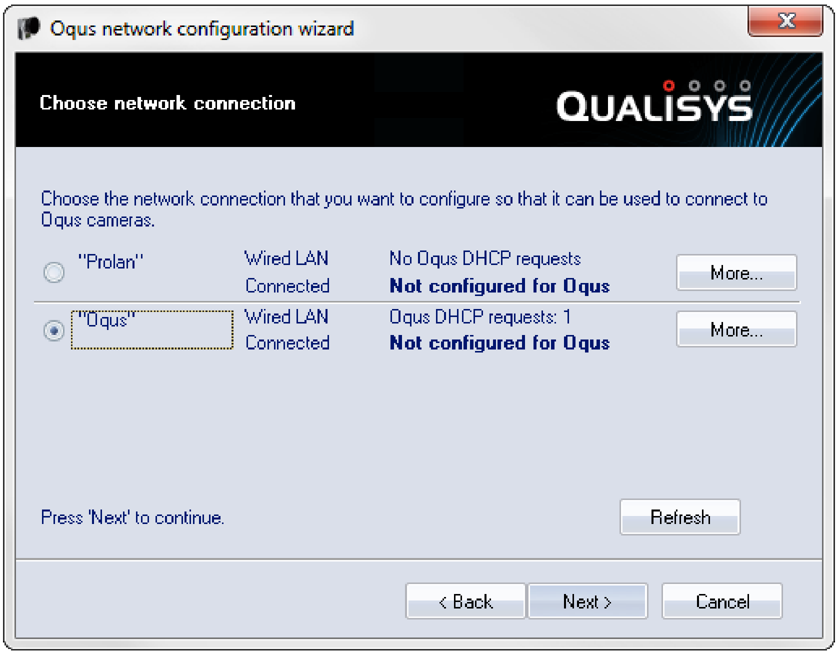
Oqus network configuration wizard

The Oqus network configuration wizard will guide you through the different steps to setup the network for Oqus cameras. If you run the wizard you do not need to follow the instructions for network card setup in chapter “Network card setup” on page 341. Follow the steps below.

1. Click on **Oqus configuration wizard** in the QDS menu to start the wizard.



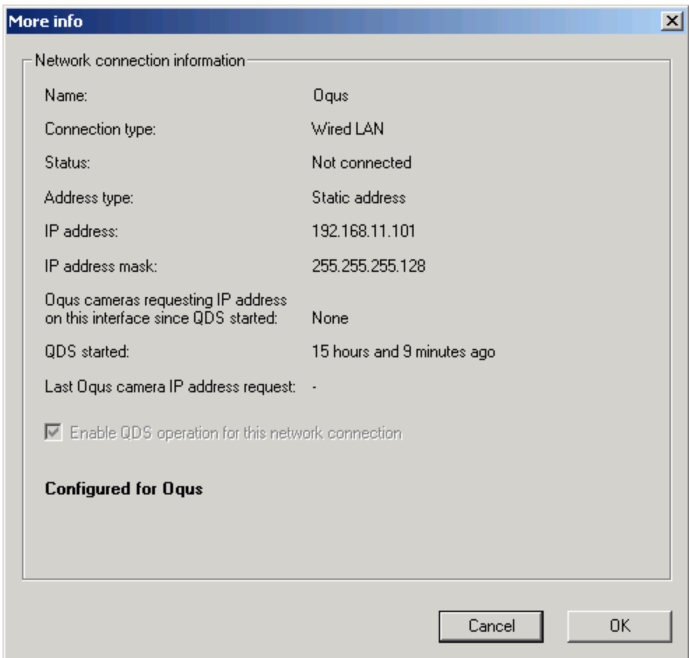
2. Select **Configure a network interface...** and click **Next**.



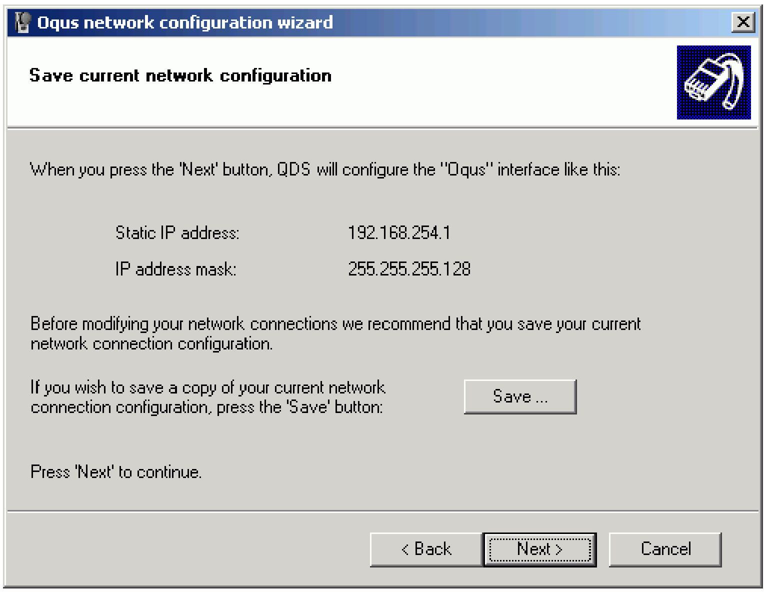
3. The list will show all enabled network interfaces on the computer. Select the interface that you want to use with Oqus cameras and click **Next**.

   Note: The wizard will not configure all interfaces, for example a network interface that is already connected to an internal network and has received an IP address will not be configured because it is considered to have a running DHCP server. However any disconnected interfaces can be configured by the wizard.

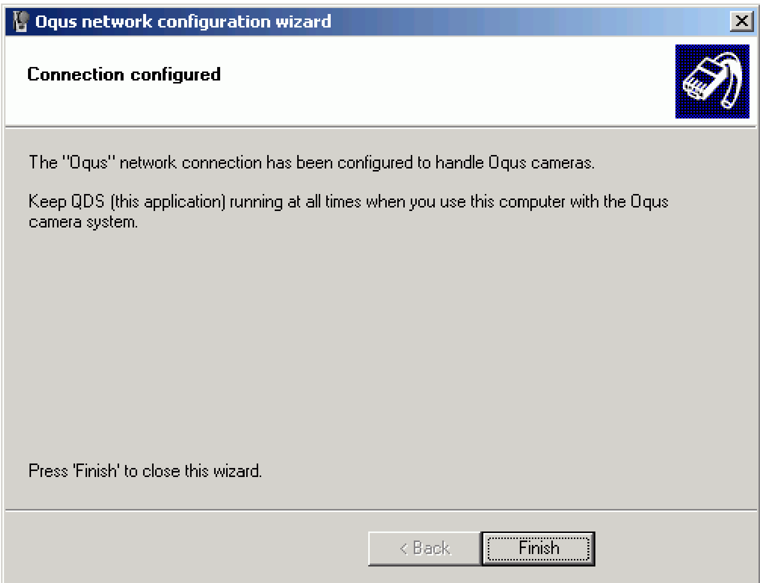
   a. Click on **More** to see information about the network interface.



More shows the current settings of the network interface. This is the same information as is shown in Advanced, see “Advanced (network settings)” on page 339.



4. The wizard shows how it will change the selected interface. You can save the current network setup with the **Save** button for backup. Click **Next** to continue.

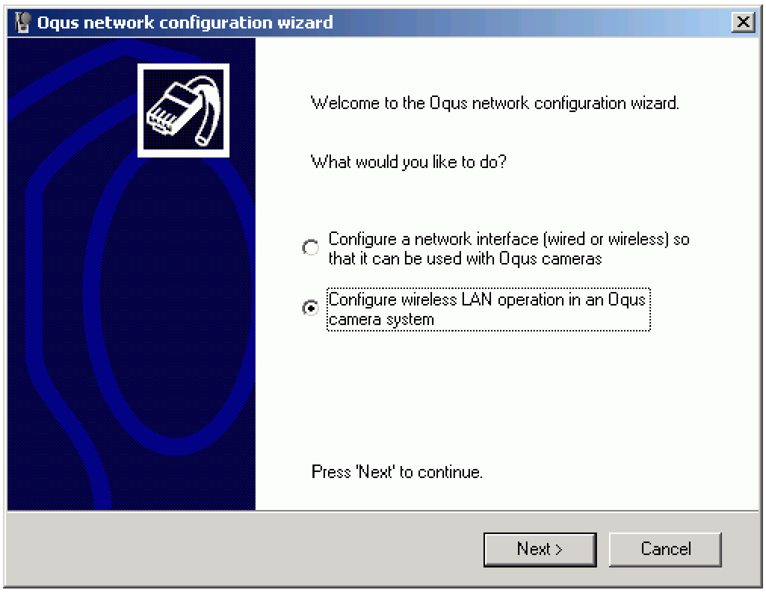


5. Click **Finish** to close the wizard.

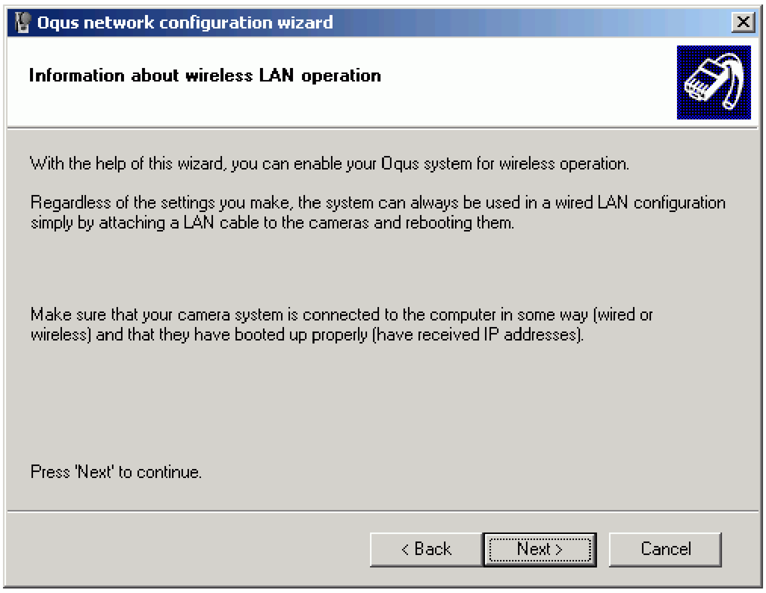
Oqus wireless camera setup wizard

The Oqus configuration wizard will guide you through the different steps to setup an Oqus cameras for wireless operation. Follow the steps below.

1. Click on **Oqus configuration wizard** in the QDS menu to start the wizard.

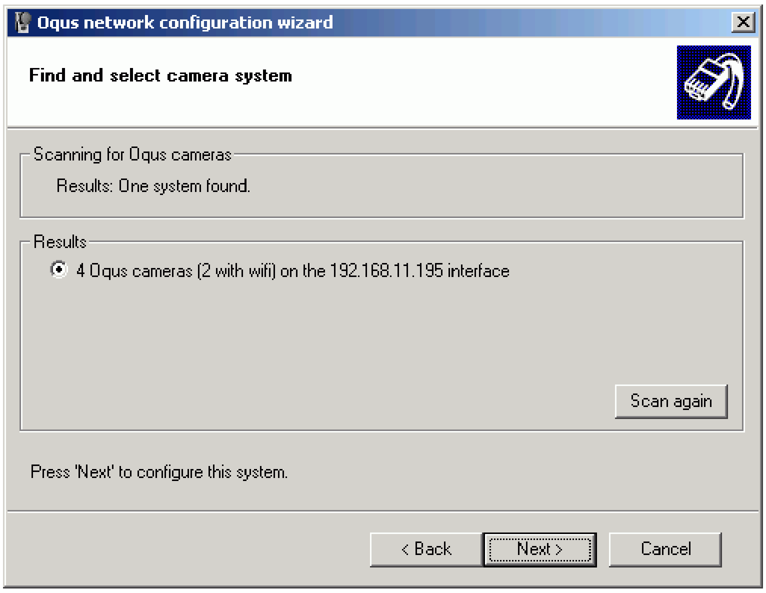


2. Select **Configure wireless LAN operation...** and click **Next**.

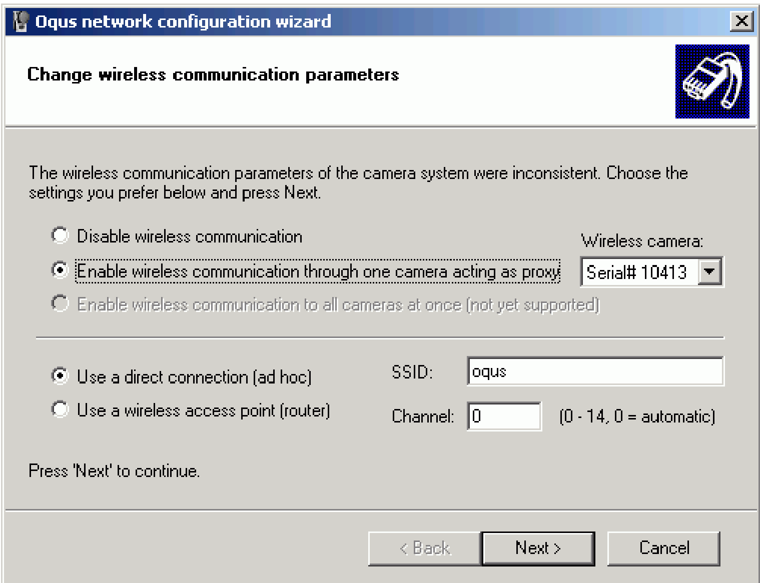


3. Make sure that the camera is connected to the computer and click **Next**.

Note: The camera can always be connected with the wired LAN even if it configured for wireless. However the wireless will only work if the camera starts up without a wired connection to the computer.



4. Select the camera system that you want to configure and click next. Use **Scan again** to scan for new camera systems.



5. The wizard will show the current settings of the camera system. Set the settings to wanted wireless setup.

   Disable wireless communication
   Select this option to disable wireless on all wireless cameras in the camera system.

   Enable wireless communication through one camera acting as a proxy
   Select this option to enable wireless communication on the camera selected with **Wireless camera**.
   Note: The wireless communication will only be activated on the selected camera, any other wireless camera will be disabled.

   Below the line is the settings for the wireless network.

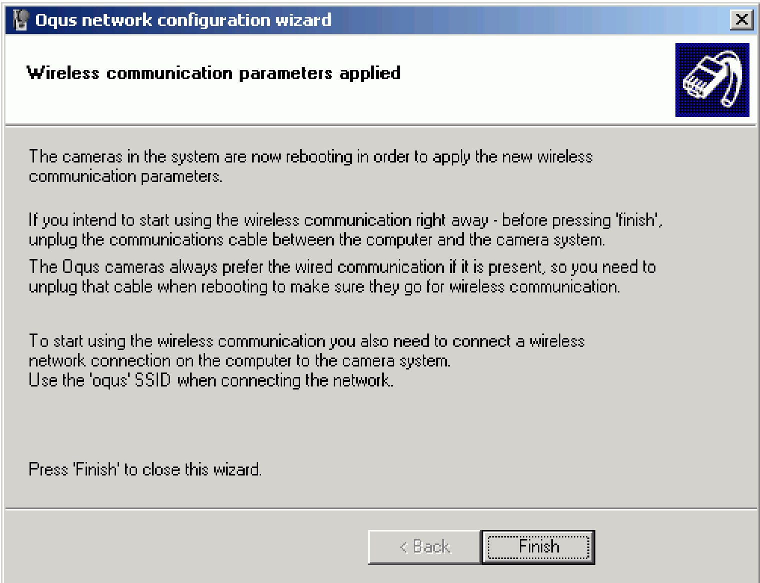
   Use a direct connection (ad hoc)
   The network is setup as an ad hoc network, which means that the camera and computer is connected directly to each other.

SSID
Choose a **SSID** so that you can identify the network. Make sure that it is not the same as an already existing network.

Channel
Select a channel for the wireless network. Use 0 for automatic if you do not know which channel you want.

Note: The **SSID** and **Channel** will default to oqus and 0 if there are wireless cameras with different wireless settings in the system.

Click **Next** to continue the wizard.



6. To make sure that the camera system starts up on the wireless interface it is important to follow the instructions in the wizard. The important steps are as follows:

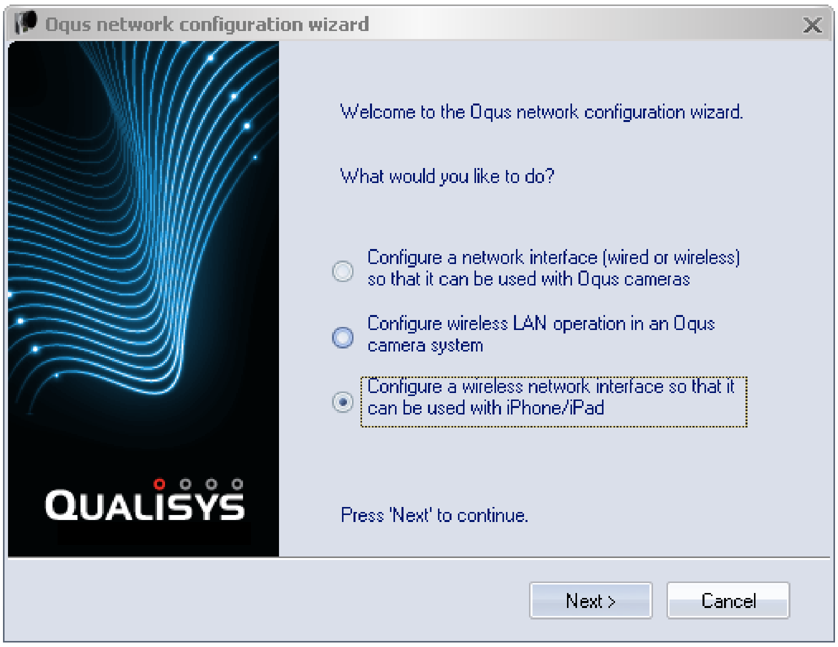
   a. Unplug the wired communication. Otherwise the camera will start on the wired interface and will not be configured correctly for the wireless communication.

   b. Then connect the wireless network on the computer to the network with the name (SSID) that was set for the wireless network. Which program to use depends on the wireless hardware, please refer to its manual.

   Click **Finish** to close the wizard. QDS will not give any new IP addresses as long as the wizard is open.

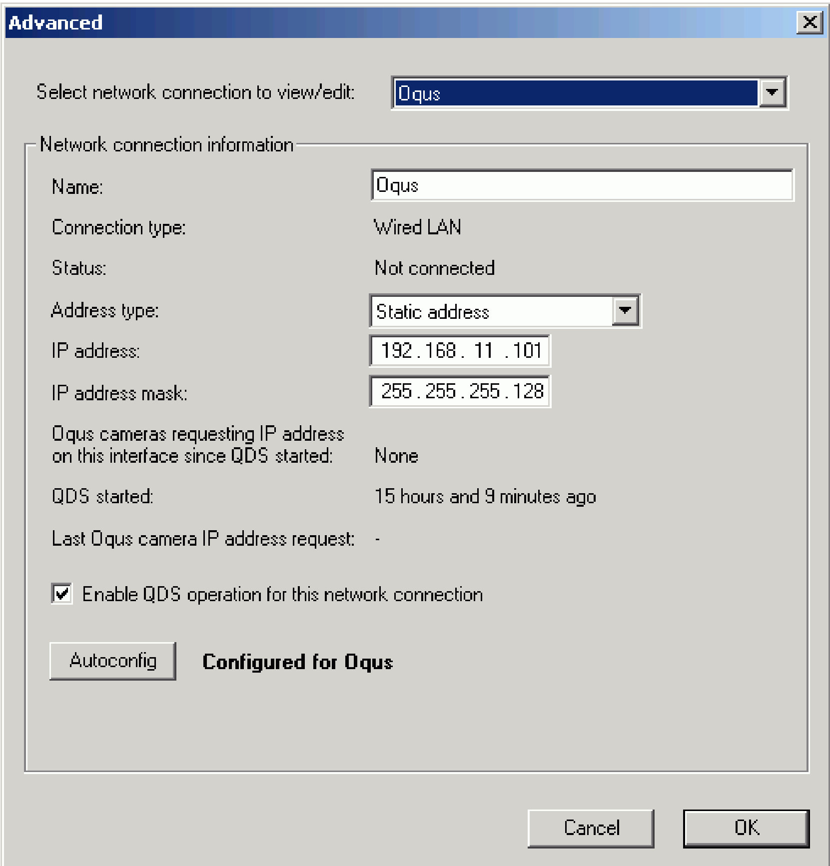
Wizard for wireless ad-hoc network for iPhone/iPad

The Oqus configuration wizard will guide you through the different steps to setup an ad-hoc network for iPhone/iPad.For the wizard to work you must first setup an ad-hoc network, see “Using an ad-hoc network for the iOS apps” on page I - 2.



Click on **Oqus configuration wizard** in the QDS menu to start the wizard. For information on how to use the wizard see “QDS settings for the ad-hoc network” on page I - 6.

Advanced (network settings)



The **Advanced** settings dialog is opened with **Advanced...** in the QDS menu. The QDS dialog contains settings for the enabled network interfaces on the computer. These settings can be used instead of the QDS wizard or the Windows network settings.

Select network connection to view/edit
Select the network that you want to edit in the dialog. The list is in the same order as in Windows and if a network is disabled in Windows network connections it will not be shown in the list.

Name
Current name of the network.

Connection type
Type of network: wired LAN or wireless LAN.

Status
Current status of the connection: Connected or Not connected.

Address type
Select the wanted address type between these two types:

Received through DHCP
The network will receive its IP address from a DHCP server. The standard setting for many networks.

Static address
The network is set to a static IP address with the settings below. This setting must be used for QDS to give IP addresses to Oqus cameras.

IP address
Current IP address of the network. The address can be changed when **Address type** is set to **Static address**.

IP address mask
Current IP address mask of the network. The mask can be changed when **Address type** is set to **Static address**.

Oqus cameras requesting IP address on this interface since QDS started
Number of Oqus cameras requesting IP address on the network.

QDS started
Time since QDS started.

Last Oqus camera IP request
Time since last camera IP request.

Enable QDS operation for this network connection
Activate QDS on the network, i.e. if QDS is disabled for the network Oqus cameras connected on that network will not get any IP address from this QDS. When networks with static IP addresses are disabled it will be shown on the QDS icon.

 Some of the networks with static IP addresses have been disabled.

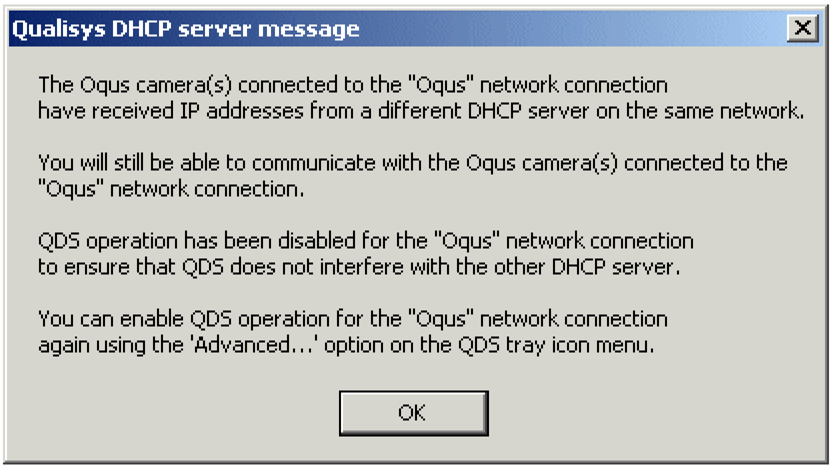
 All of the networks with static IP addresses have been disabled.

Autoconfig
Use this button to configure the network interface for Oqus cameras.

QDS conflict

There will be a conflict for the IP addresses when two or more computers with QDS are connected to the same camera system. In those cases the first QDS that replies after the startup of a camera will give the IP address to the camera.

On the other computers the Qualisys DHCP server message below will be shown and QDS operation will be disabled on that network. The QDS operation can be turned on again with the **Advanced** option on the **QDS** menu. However, make sure that the other computers are disconnected from the Oqus system otherwise QDS operation will be turned off again at the camera startup.



Network card setup

It is recommended to use a computer with two network interfaces, one reserved for the Oqus system and one for an office network or internet. This could be either two Ethernet interfaces or one Ethernet and one WLAN. If you only have one interface it should be dedicated to the Oqus system.

IMPORTANT: Do not connect the Oqus system to wired USB to ethernet adapters. The communication cannot be guaranteed on the adapters and there can be communication errors when capturing data.

The recommended setup is to use static IP on the Oqus network, other setups are possible but not recommended by Qualisys. Use the wizard to setup the network interface, see “Oqus network configuration wizard” on page 333.

To setup the network interface manually with a static IP-address follow the steps below.

1. Before changing the network configuration you can save the current configuration in QDS, if you want to use it later. This is recommended when you only have one network card.
2. Click on the Windows **Start Menu\Settings\Network Connections** to open **Network Connections** window.
3. Select the network card that you want to use for the Oqus system. Right-click on the card and select **Properties**.
4. In the list locate the item **Internet Protocol**. Select it and click on **Properties**.
5. Click on **Use the following IP address**.
6. Enter an **IP address**. It must start on 192.168.

   Wired LAN
   It can for example be 192.168.20.1. It is important to check that the first three numbers are not already used on an existing interface. Then the fourth number must be less than 128.

   Wireless LAN
   It can for example be 192.168.20.129 is important to check that the first three numbers are not already used on an existing interface. Then the fourth number must be more than 128.

Note: If there are more than one computer that is connected to the Oqus wireless network they cannot have the same IP address.

7. Then enter the **Subnet mask** as 255.255.255.128 for Wired LAN and 255.255.255.0 for Wireless LAN. Click OK twice.

Setup Oqus system for wireless communication

The Oqus system can run with a wireless communication from the camera system to the computer. The camera uses the 802.11b/g@54mbps standard. However the communication speed can be reduced depending on the signal strength or if there are many other wireless networks.

Follow these steps to enable the wireless communication

1. Run the **Oqus network configuration wizard** to setup the wireless network interface on the computer for the Oqus system. Follow the instructions in “Oqus network configuration wizard” on page 333.
2. Make sure that the wireless network on the computer is not connected to any wireless network. Also make sure that no wireless network is set to automatic connection.
3. Connect the cameras to a wired network on the computer and run the QDS wizard to enable wireless in one of the cameras. Follow the instructions in “Oqus wireless camera setup wizard” on page 335.
   It is important to connect the wireless network on the computer to the Oqus network when it is created. Otherwise QDS will not give the cameras IP addresses.

Setup Oqus system with Ethernet switch

The Oqus system can be setup in a daisy-chain, each camera connected to a Ethernet switch or in any combination of those setups. For most systems the recommended the setup is the daisy-chain as described in “Setting up the system (Oqus)” on page 329. However if the camera system is used a lot for high-speed video capture or if there are more than 16 cameras in the system, then it can be better to use a Gigabit Ethernet switch. It must be a Gigabit Ethernet switch to be able to handle more data than a daisy-chained system.

To setup the system with the Gigabit Ethernet switch follow these instructions:

1. Check that the computer has a Gigabit Ethernet card. Otherwise using a Gigabit ethernet switch will not change the performance of the system.
2. Connect the Ethernet card with the switch. You can use any of the ports on the switch.
3. Use the standard network card settings as described in the chapter “Network card setup” on previous page.
4. Connect the cameras to the ports on the switch using host cables. There can be any number of cameras connected in a daisy-chain to each port. But the following are the two main setups.

   Setup for high-speed video
   To maximize the video fetching time it is best to have daisy-chains of only 2-3 high-speed video cameras connected to each port. Note that the fetching time is not improved much by just having one camera at each port compared with two.

   Setup for more than 16 cameras
   Connect the cameras in daisy-chains of a maximum of 12 cameras. It is

of course best to have a similar amount of cameras on each chain, e.g. a 28 camera system can have three chains with 10, 9 and 9 cameras.

The two setups can be combined, e.g. you can have 8 marker cameras in a daisy-chain connected to one port and then 4 high-speed video cameras split up in two chains connected to two other ports.

Note: Do not connected the different daisy-chains to each other. It will mess up the communication and the switch will be of no use.

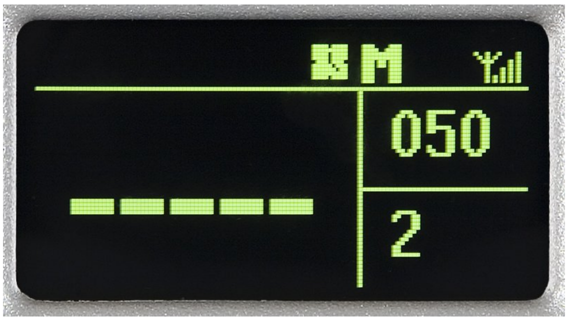
Oqus startup sequence

This is the general Oqus startup sequence.The Oqus camera must be connected to a computer with a QDS running during the startup. This is because the camera must receive an IP-address from QDS to be able to communicate with other Oqus cameras. For more information see “QDS” on page 332.

1. Connect the power supply and the green LED on the front will blink twice.
2. After a few seconds the startup bar below is shown.

If the bar stops at two-thirds then the Oqus is waiting for an IP-address. The reason is probably either a missing connection to the computer or that QDS is not running, for instructions on how to search for the error see “Troubleshooting connection” on page 387.

3. When the camera has an IP-address the display will show an image similar to one below. The Oqus will first synchronize to other cameras, during that process the clock is blinking and there is a spinning plus sign instead of the letter M or S. Wait until the clock stopped blinking and the display shows M or S.



The M or S on the display stands for Master respectively Slave. This is only to show which camera that is sending a synchronization pulse to the other cameras.

Setting the aperture and focus

For Oqus it is very important to set the aperture and focus correctly for your measurement volume. If these are not set correctly the cameras will not reach their full potential in detecting markers. Follow this procedure to set the aperture and focus.

1. Turn the strobe part of the camera counterclockwise to expose the lens for adjustment.
2. Open the aperture as much as possible, i.e. set the lowest aperture.
   Note: For the Oqus 1 series and 3+-series it is sometimes possible to have the aperture on a higher number than the lowest. Check this by using the **Marker intensity** mode in QTM to see whether the markers are bright enough. The advantage with a more closed aperture is a longer focal depth.
3. Place a marker in the measurement volume. Use the same size as will be used in the actual measurement.
4. Switch the cameras to **Video** mode in QTM and change the focus until the markers are as small and sharp as possible. Make sure that the **Flash time** is long enough on the **Camera settings** page.
   a. a.When you have set the focus correctly, switch to **Marker intensity** mode in QTM to make sure that the markers are visible in **Marker** mode. If the color of the markers is not red, set a longer exposure time in QTM on the **Camera settings** page in QTM.
5. Turn the strobe back (clockwise) to close it. The strobe should always be closed during measurement, to achieve the best strobe light distribution.

Arranging the cameras

Once the system has been properly set up, the cameras must be arranged to the current measurement setup. When arranging the cameras, it is best if they are in operating mode.

1. Start the measurement computer and the QTM software.
2. Locate the camera system in QTM on the **Connection** page in the **Project options** dialog, see “Camera system connection” on page 227.
3. Open a new file with a **2D view** window.
4. Arrange the cameras to cover the entire measurement volume of the wanted motion. It is suggested to mark the corners of the measurement volume with markers, and use the **2D view** window to make sure that each camera can see the markers as expected. The **Video** view can also be used to see the camera field of view. For guidelines on how to arrange the cameras see “Camera positioning” on page 250.

Note: The number of markers seen by each camera is also shown on the display on the front of each camera.

Mixing Oqus camera types

The Oqus types are compatible so that different Oqus types can be mixed in one system. The connection is done in the same way as for a regular Oqus system and the cameras can be placed in any order you wish.

When mixing camera types the global camera settings will default to the lowest of the camera limits. This means for example that the capture rate will be limited to 187 Hz at full field of view, when 3- and 5-series are mixed. However individual settings can always be set within the limit of the camera type.

Oqus 3+ features compared to other Oqus models

The Oqus 3+-series has a new sensor with some new features and improvements which are all described below.

Light-sensitivity
The Oqus 3+ is at least 50% more light-sensitive than the other Oqus models. This means that you can see markers at a longer distance with the same exposure and threshold setting as on the 3-series. Or you can use smaller markers at the same distance as you could for the 3-series.
Note: The default exposure time setting for Oqus 3+ is 200 microseconds, because of the increased light sensitivity.

The increased light sensitivity also means that it is easier to see the markers at higher frequencies, because you can use a shorter exposure time.

In small volumes, where the cameras are about 3 m from the subject, the aperture can sometimes be closed more than 2.8. Use for example 4 or 5.6. The advantage is longer focal depth and therefore sharper markers.

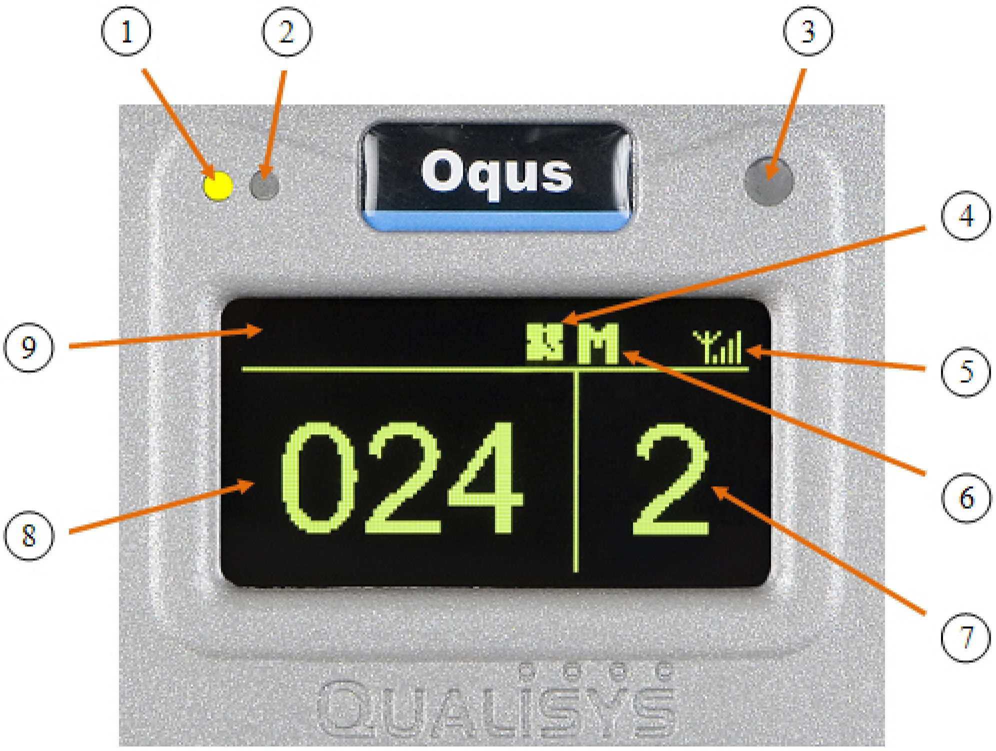
Improved image
The image has less static and dynamic noise compared to the other cameras. Therefore the marker threshold can often be lower than on the other camera types.

Sensor High-speed mode
The Oqus 3+ has a special sensor mode where you only use a fourth of the pixels but still keep the field of view. The camera can then capture at 1764 Hz with the sensor resolution of 648*512, slightly larger than the 1-series resolution. For more information see “Sensor mode” on page 155.

Active filtering
The Oqus 3+ camera supports active filtering, in which each image is captured twice to remove the background light. This dramatically increases the ability to capture passive markers in an outdoor environment. For more information see “How to use active filtering” on page 264.
Note: Also available on the 1-series cameras.

Oqus display

The Oqus camera has a large graphical OLED display and three LEDs on the front to inform the user of the current status of the camera. The display shows, among other things, the camera number and the number of markers currently seen by the camera.
Note: The display will be turned off when the camera enters stand-by mode, i.e. if the camera has not been in use for 2 hours. Start a preview in QTM to light up the display again.



1. **Measurement status indicator**
   Green light - The camera is ready to start a measurement
   Yellow light - The camera is measuring
   Flashing green light - Waiting for trigger to start measurement
   Flashing yellow light - Waiting for trigger to switch from pre-trigger to post-trigger measurement

2. **Error indicator**
   A red light indicates that an error has occurred. The LED is blinking when a software error occurs and is lit constantly if a hardware error occurs.

3. **IR receiver**
   The IR receiver is used for synchronization with certain active markers. It detects modulated light with a frequency of 33 kHz and is sensitive to light with wavelengths between 800 and 1100nm.

4. **Synchronization status**
   During the synchronization phase this symbol is flashing. When the camera is synchronized with the master camera in the system it becomes stable.

5. **WLAN indicator**
   This symbol is displayed when the WLAN of the camera is activated.

6. **Master/Slave indicator**
   An M indicates that the camera is master for the system and by that controls for example internal synchronization. An S indicates that the camera is a slave. The indicator can also be a rotating + sign, which means that the camera is looking for the Master camera.

7. **Camera number**
   The area to the right usually shows the camera number that the camera has in QTM. The camera number can be changed with the **Reorder** tool in the 2D view window. This number is stored in the camera so it is shown at the next camera startup.
   Note: If the camera has never been connected to QTM the last three digits

of the serial number (upper part) and the last octet of the IP-number assigned to the camera (lower part) will be shown instead. This can also be activated from the QDS menu, see “QDS” on page 332.

8. **Marker area**
   During a marker measurement this area shows the number of markers currently seen by the camera. When the camera is idle or is collecting video, this area shows ’-----’.

9. **Text area**
   This area is used for scrolling text messages, for example during startup.

Oqus high-speed video camera

The high-speed video version of a Oqus camera is adapted to capture full-frame, full-speed, full-resolution high-speed video. In this configuration the camera is therefore equipped with a large buffer memory and a clear front glass to get the best possible performance out of the image capture.

The clear front glass is mounted so that all of wavelengths can be captured by the camera. The normal dark front glass is an IR-filter that removes the visible light. However the high-speed version is also delivered with a removable IR-filter on the lens. Which is important to mount if the camera is in the marker mode, because the data is improved when the visible light is removed to increases the contrast between the background and the marker. For instructions on how to get access to the lens see “How to change strobe unit” on page A - 7.

Oqus underwater system

Qualisys provides the possibility to measure under water, (e.g. in indoor ocean basins used for ship scale model testing or ordinary basins, used for water sports) by using specially modified Oqus motion capture cameras.

Oqus underwater cameras are equipped with a special strobe with high power cyan LED:s. These LED:s are not limited to a flash time of 10% of period time as the regular Oqus LED. Therefore the exposure time can be set to almost the period time. The long exposure times are needed to get enough light in the water. Because the water absorbs more light than air it also means that the measurement distance are more dependent on the exposure time.

Warning: Be careful when using the camera out of water and do not use longer exposure times than 2 ms.

The FOV is also changed by refraction index of water (1.33). This means that a lens with 40º FOV is reduced to 31º FOV.

ProReflex

The ProReflex MCU has four different models: 120, 240, 500 and 1000. The difference between these is the maximum capture rate, but they can be used together in the same system. The chapter “Setting up the system (ProReflex” below describes how to connect and set up the system.

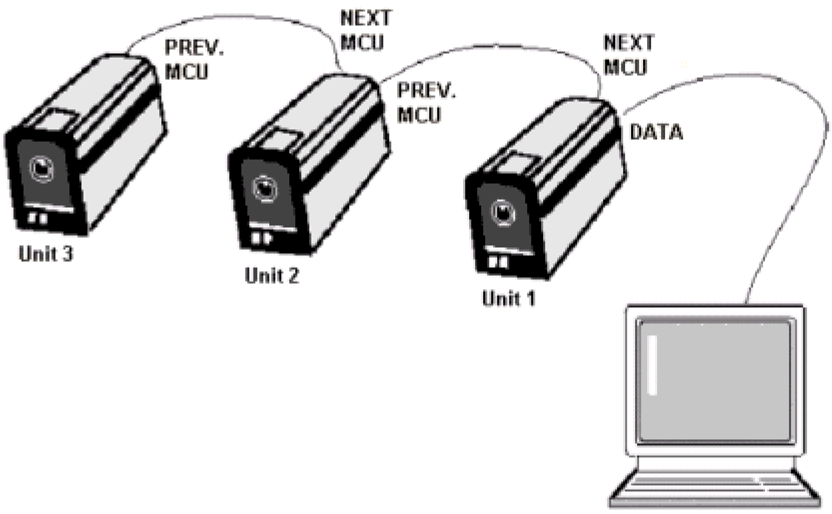
Setting up the system (ProReflex

This chapter is a description of how to connect a system with ProReflex MCUs.

Up to 16 MCUs can be used in a 3D motion capture system, where the units are connected to each other with the **Next MCU** and **Prev. MCU** ports. The entire camera system is then connected to the measurement computer with the **Data** port of the master MCU, i.e. the first MCU in the chain. Use the cables that are distributed with the system to connect the system.

To set up the system, go through the following steps:

1. Connect the data cables (blue) between the MCUs. The IDs of the cameras can be set in any order.
2. Connect the RS 422 cable (yellow) between the **Data** port of the master MCU and the RS 422 port on the serial communication board in the measurement computer.
   Note: If a regular COM port is used connect the RS 232 cable (black RJ45 connector) between the **Data** port of the master MCU and the COM port.
3. Set all power switches to off and connect all units to power supplies.
4. Switch on the power supply of the master MCU. All other units will then be automatically switched on.



Arranging the cameras

Once the system has been properly set up, the cameras must be arranged to the current measurement setup. When arranging the cameras, it is best if they are in operating mode.

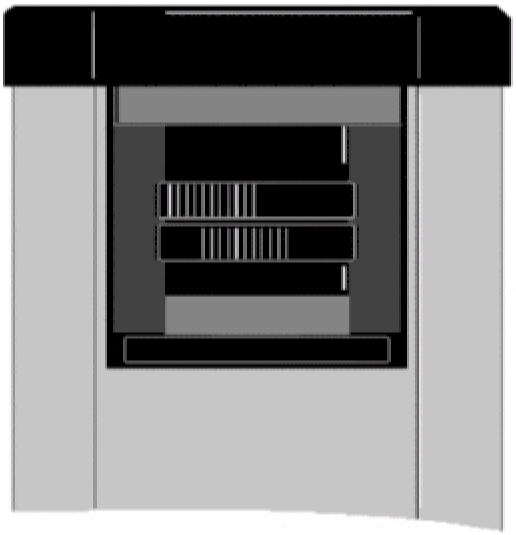
1. Start the measurement computer and the QTM software.
2. Locate the camera system in QTM on the **Connection** page in the **Project options** dialog, see “Camera system connection” on page 227.
3. Open a new file with a **2D view** window.
4. Arrange the cameras to cover the entire measurement volume of the wanted motion. It is suggested to mark the corners of the measurement volume with markers, and use the **2D view** window to make sure that each camera can see the markers as expected. The **Video** view can also be used to see the camera field of view. For guidelines on how to arrange the cameras see “Camera positioning” on page 250.

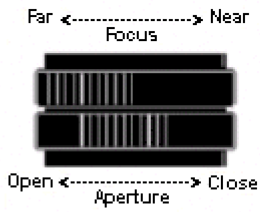
Note: The number of markers seen by each camera is also shown on the display on the front of each camera.

Aperture and focus

The lens of the MCU can be reached through the access door at the top of the unit. Two parameters of the lens can be adjusted: aperture and focus. It is recommended to first set the focus and then the aperture according to the following sections. The focus is set on the adjustment ring closest to the front, while the aperture is set on the one closest to the rear.

Important: Before adjusting aperture and focus of the cameras, the frequency of the measurement must be set. This is done in the QTM software on the **Camera system** page in the **Project options** dialog. The reason for setting the frequency first is that the camera resolution and light sensitivity are frequency dependent.



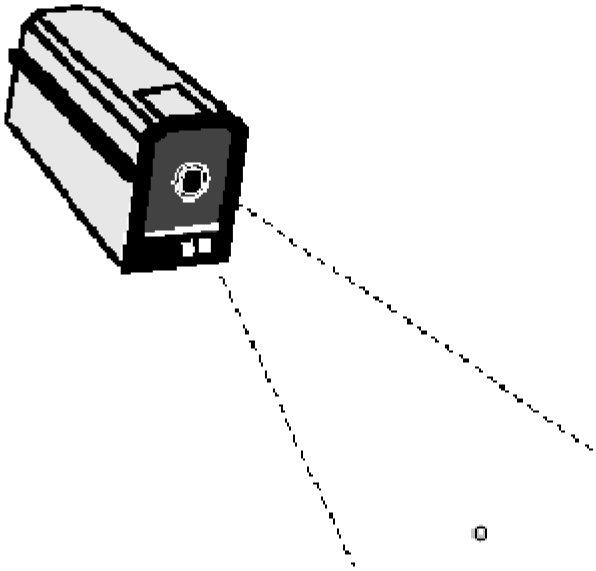


Setting the focus

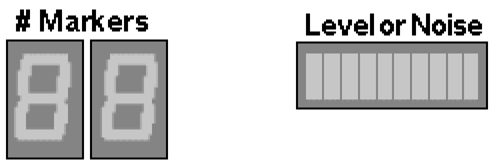
When the camera system has been arranged, the focus of each camera is set by going through the following steps:

1. Make sure that the correct frequency is set for the camera system and place a small marker at the furthest possible distance from the camera, within the measurement volume. In the table below, suitable marker sizes for setting the focus are shown for different distance ranges.

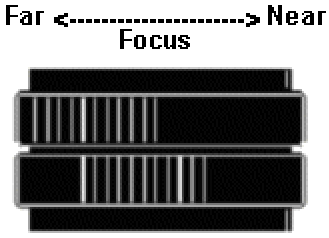
Marker size	Distance range
4 mm	< 1m
7 mm	1 - 2 m
12 mm	2 - 4 m
19 mm	4 - 6 m
30 mm	6 - 8 m
40 mm	> 8 m

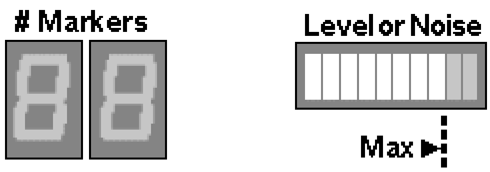


2. Adjust the aperture until the **# Markers** display shows '1' marker and approximately seven to eight bars of the **Level** indicator are filled.



3. Adjust the focus ring until the optimum focus is achieved. It is when the **Level** indicator reaches its maximum value for this setup. The maximum level is therefore not equal to 10 bars on the **Level** indicator.

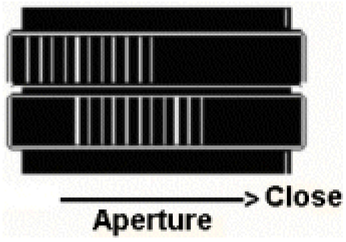




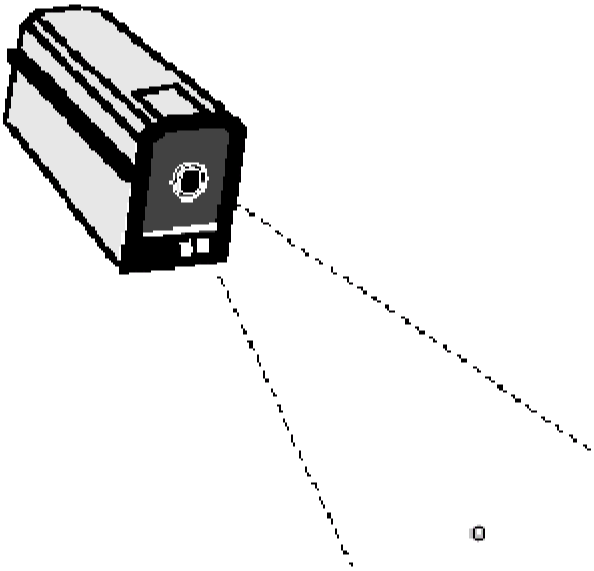
Setting the aperture

After the focus has been set, the aperture of the camera is set by the following procedure:

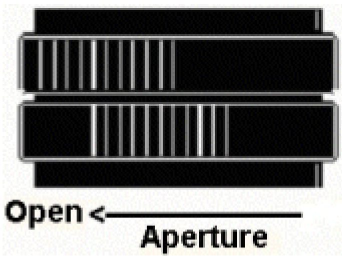
1. Close the aperture completely by turning the aperture ring. No bars on the **Level** indicator will be lit.

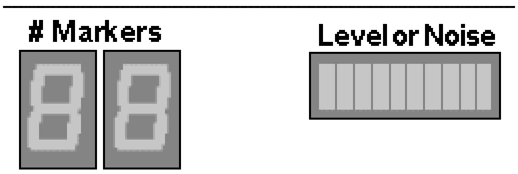


2. Place a marker, with the same size as those that will be used in the measurement, at the furthest possible distance from the camera, within the measurement volume.



3. Open the aperture until the # **Marker** display shows '1' marker and approximately eight bars of the **Level** indicator are filled.





MacReflex

MacReflex cameras can be used with QTM 2.0.387, if you have a SeaLevel board with the correct cable. However the hardware support on MacReflex related problems is limited because it is an outdated product.

MacReflex in QTM

Post-processing of files with MacReflex data will always be supported in QTM, so you can always process data captured by MacReflex in the latest version of QTM. However 2.0.387 is the last QTM version where the capture of data from MacReflex is tested by Qualisys AB. It may work in future versions, but we will not do any fixes if the capture is broken.

QTM 2.0.387 supports the 60 Hz version of MacReflex (VP II, type 170 002) with Sealevel PCI ULTRA 530.PCI [7101] board. To connect the system the cable 190115 from Qualisys must be used.

MacReflex do not use the **Flashes** options and it is not possible to use **Real time processing while capturing**. However it can use RT output in RT/preview mode, but the frequency will be limited because of the limited communication speed.

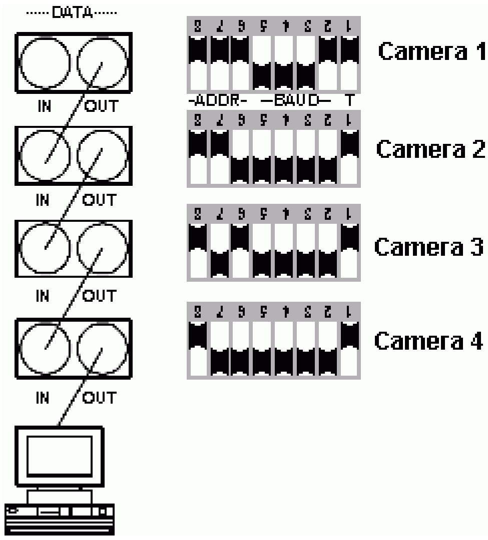
Because MacReflex runs at a constant frequency there can sometimes be strange numbers in the status bar, but it always runs at the divisor of 60Hz set on the **MacReflex** page. For example in a calibration if the calibration time is set to 10 s, the MacReflex will run at 30 Hz so the calibration time will in fact be 16.67 s.

Connecting the MacReflex system

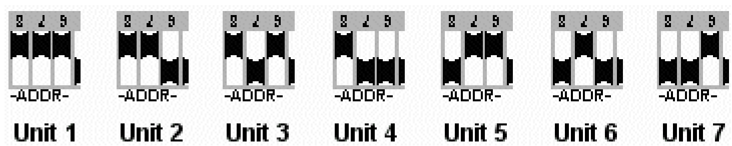
Follow these instructions to connect the MacReflex system. For more detailed instructions of the MacReflex hardware see the MacReflex manual.

1. Check that the jumpers and the settings for the Sealevel board are correct, see “Serial communication devices - Technical specification” on page C - 1.

2. Connect the system and set the switches according to the image below. The example uses a baudrate of 500000. If there are problems with that baudrate you can try 250000 or 19200, see MacReflex manual. For example for six cameras it can be good to lower the baudrate to 250000.
   Note: QTM will always report 500000 as baudrate when you locate the system, but it runs at the frequency that you have set on the camera 1.



If you have more than 4 MacReflex cameras set the switches for ADDR according to the image below.



3. Locate the system and then set the frame rate for MacReflex cameras on the **MacReflex** page in the **Project options** dialog.

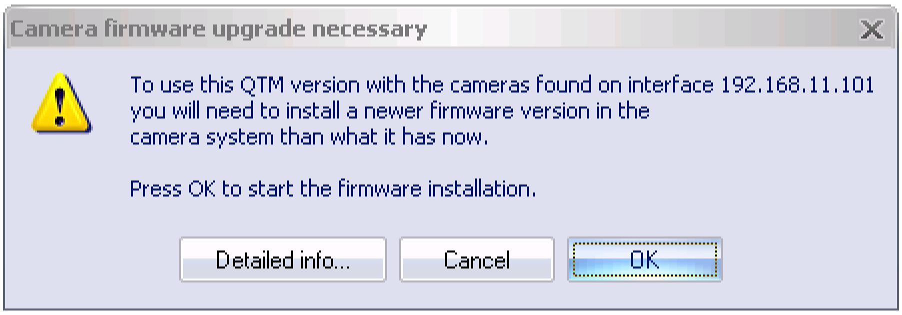
Firmware update

A certain version of QTM is built to work with certain versions of MCU firmware. Therefore a camera system cannot be used if the firmware does not have the same major revision as QTM is adapted to. E.g. QTM can be built to work with firmware version 7.03. Then QTM will not work with firmware versions 7.02 or 7.04, since the number '2' and '4' is the major revision number.

QTM will detect automatically if the camera firmware needs to be updated or downgraded. The detection is made either when you locate the system on the **Connection** page in the **Project options** dialog or when you open a new file with **New** on the **File** menu.

Firmware update when locating system

The following dialog will appear when QTM has detected an old firmware when you are locating the system on the **Connection** page in the **Project options** dialog.



Detailed info...
Open a dialog with information on why QTM needs to update the firmware.

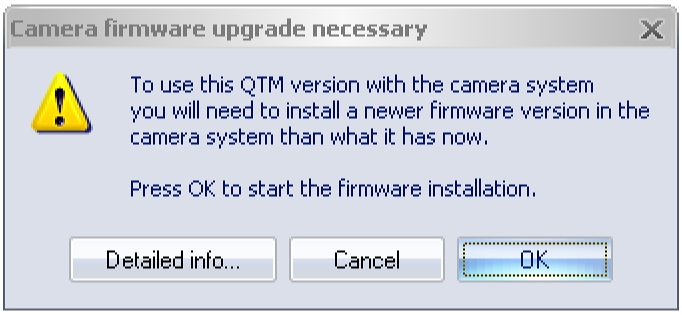
Cancel
Cancel the update. However, you **cannot** use the system with the current QTM version until you have updated the firmware.

OK
Start the firmware update program QFI.exe. For more information on QFI.exe see “Qualisys Firmware Installer (QFI)” on page D - 1.

Note: If the current camera firmware is a higher major version than QTM is adapted to, the firmware will have to be downgraded instead.

Firmware update when opening a new file

The following dialog will appear when QTM has detected an old firmware when you are opening a new empty capture file with **New** on the **File** menu.



Detailed info...
Open a dialog with information on why QTM needs to update firmware.

Cancel
Cancel the update. However, you **cannot** open a new file until you have updated the firmware.

OK
Start the firmware update program QFI.exe. For more information on QFI.exe see “Qualisys Firmware Installer (QFI)” on page D - 1.

Note: If the current camera firmware is a higher major version than QTM is adapted to the firmware will have to be downgraded instead.

Serial communication boards

Supported serial communication boards

The Qualisys Track Manager software uses serial communication devices of the measurement computer to communicate with the camera system. QTM can use the following devices for the communication:

- Built-in COM ports
  Max baud rate 115.2 kbit/s

Windows XP

- Brainboxes PCI Dual Port RS422/485 [CC-530] board
  Max baud rate 2000 kbit/s
- Brainboxes PCMCIA Single Port RS422/485 [PM-120] board
  Max baud rate 921 kbit/s
- Sealevel PCI ULTRA 530.PCI [7101] board
  Max baud rate 2048 kbit/s
- Sealevel Low Profile PCI ULTRA 530.LPCI [7106-SN] board
  Max baud rate 2048 kbit/s
- Sealevel PCMCIA ULTRA-PC-SIO-485 [3602-SN] board
  Max baud rate 921 kbit/s

Windows 7

- Brainboxes PCIe 1xRS422/485 1MBaud Opto Isolated [PX-387] board
  Max baud rate 921 kbit/s
- Brainboxes ExpressCard 1xRS422/485 1MBaud [VX-023] board
  Max baud rate 921 kbit/s

To avoid long download times from the camera system it is recommended that one of the communication boards is used, at least when there are more than two cameras in the system.

Use the drivers supplied with the QTM installer for the Brainboxes and Sealevel boards. Other drivers have not been tested with QTM and may therefore not work. Sealevel have drivers for Windows 7, but these have not been extensively tested. And because Sealevel have not got ExpressCard for the laptop, Qualisys will only support Brainboxes on Windows 7.

For information about how to install the different boards see “Serial communication devices - Technical specification” on page C - 1.

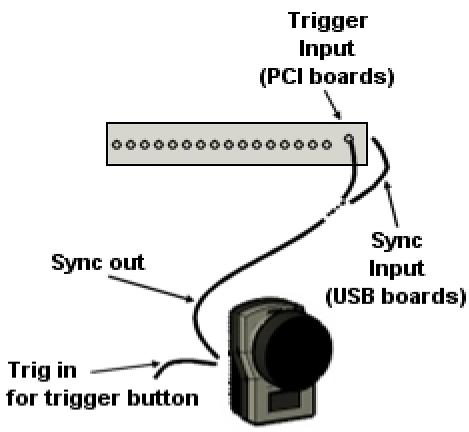
How to use analog boards

Connection of analog board

An analog board is needed in the measurement computer to capture analog signals together with the motion capture. For information on how to install an analog boards see “Analog capture hardware” on page E - 1.

The analog board is activated on the **Analog boards** page in the **Project options** dialog, see chapter “Analog boards” on page 161. It is important to activate channels, where a cable is connected, on the page for the active analog board in the **Project options** dialog. It is also good idea to not use the channels where nothing is connected, since these channels will only make the file larger.

To get synchronized analog data and motion capture data a synchronization signal must be sent to the analog board. The connection differs some between the camera systems and analog boards.



USB-2533 and USB-1616FS
For Oqus the recommended synchronization option is the frame synchronization, see chapter “External sync” on page 164. This option can also be used with ProReflex but then the analog frequency is always the same as the camera frequency. For this option you must connect the **Sync out** connection from the camera to the **Sync** connection on the back of the box.

USB-2533
For USB-2533 the trigger edge that is used by the board must be set in Instacal see chapter “Installing the USB-2533 board” on page E - 6. Make sure that the **XAPCR edge** setting for the board matches the **TTL signal polarity** setting on the **Advanced (Timing)** page in the **Project options** dialog.

USB-1616
USB-1616 always trigger on a rising edge, which means that the **TTL signal polarity** setting must be set to **Positive**.

Warning: Because the **Sync** port on the USB-1616 is used for both synchronization options it is important that the **Use external sync** setting is not deselected without removing the connection from **Sync out** on the camera to **Sync in** on the analog board. If the connection is not removed it can damage the equipment.

The other option is to start the analog capture with the sync output from the camera. Then the frequency is set internally by the analog board, which means that there may be a small drift between the systems. This option must be used if you want a higher frequency on the ProReflex than the camera frequency. To use this option connect the **Sync out** connection from the camera to the **External trigger** connection on the front of the box.

PCI-DAS6402/16 and PCI-DAS1602/16
For the PCI board the only synchronization option is to start the analog capture with the sync output from the camera. Therefore the synchronization cable is connected between the **Sync out** connection of the camera system and the **External trigger** connection on the analog board. This connection is the BNC port furthest to the right on the front of the box.

Note: On a ProReflex system the control port cable must always be connected to the Master camera. For Oqus it is recommended to connect the cable to the Master camera if the synchronization signal is faster than 5000 Hz.

Note: For the PCI analog boards it is recommended to use a BNC connection box for easier to connection of equipment to board, see chapter “BNC connection box” on page E - 15. If you don't have a BNC connection box please refer to the manual from Measurement computing for details on the connection.

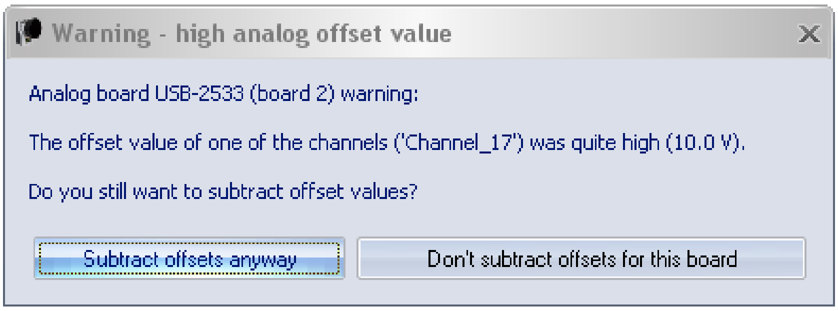
The number of analog samples (frames) depends on the **Sample rate** setting on the page of the analog board in the **Project options** dialog, see chapter “Board settings” on page 162. Therefore the number of analog frames does not have to be the same as the number of frames in the motion capture.

Another setting on the page for the analog board in the **Project options** dialog is the remove offset and drift option. Offset is for example that a force plate has an output that differs from 0 V when it is unloaded. Drift is for example when the output from the force plate slowly changes even when it is unloaded. There is a warning in QTM if the offset value is higher than 5 V, see chapter “”.

Analog offset warning

QTM will warn you if the offset that is being removed from a channel is higher than 5 V. This is because it usually indicates that there is more offset than you would usually like to remove. For example if you stand on a force plate when the measurement starts, then if you remove the offset the force data will be wrong.

The offset compensation is activated for each analog board on its respective page, see chapter “Compensate for analog offset and drift” on page 163. The warning is always displayed at the end of a measurement if the offset is too high.



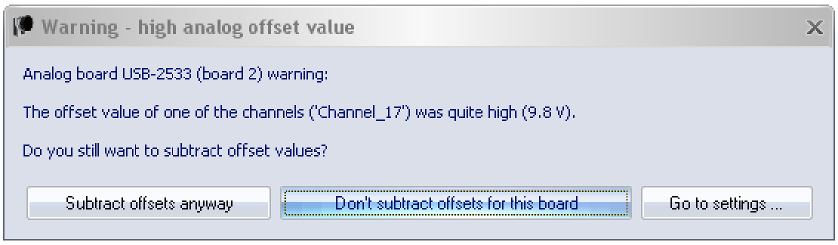
You have the following two options in the warning after you have captured a file.
Note: If you have selected the **In RT too** option, the warning will appear directly when you start the capture. The capture will however continue and if you like you can

wait to choose what you want to do until the capture is finished. If you choose what to do before the file has finished, the warning will not appear again.

Subtract offset anyway
Subtract the analog offsets on the board in the file and keep the offset compensation options activated on the **Analog board** page.

Don't subtract offsets for this board
Do not subtract the offsets on the board in the file and turn off the offset compensation options on the **Analog board** page.
Note: If you do not want to turn off the offset compensation options, select **Subtract offsets anyway** and then right click in the **Data info window** to turn off the offset compensation on the channels with too high analog offset, see chapter "Analog data information" on page 82.

If the **In RT too** option is activated on the **Analog board** page, then there is also a warning when you start RT/preview with **New** on the **File** menu. If you do not use the RT output it is recommended to inactivate the **In RT too** option, because for example if you zero a force plate after you started RT/preview then you will get the wrong force data during RT. However the data in a captured file will still be ok, because the offset is calculated again from the first samples in the file.



You have the following options when the warning is displayed in RT/preview when you start a new measurement with **New** on the **File** menu.
Note: The offset check is performed every time the RT has to be restarted, for example if you change a setting in Project options.

Subtract offset anyway
Subtract the analog offsets on the board in RT and keep the offset compensation options activated on the **Analog board** page.

Don't subtract offsets for this board
Do not subtract the offsets on the board in RT and turn off the offset compensation options on the **Analog board** page.
Note: If you do not want to turn off the offset compensation for all of the channels, select **Go to setting...** and then select on which channels to use the offset compensation, see chapter "Channels" on page 165.

Go to settings...
Open the **Project options** dialog and go to the **Analog boards** page. You have to select the correct analog board manually to change the offset settings, see chapter "Compensate for analog offset and drift" on page 163.

Analog devices

Any analog device which has an output voltage between ± 10 V can be connected to the analog board. QTM will however only make calculations on data from force plates. For information about force plates and EMG devices see chapters "How to use force plates" on the facing page and "How to use EMG" on page 364.

How to use force plates

Connection of force plate

With a force plate the force can be captured together with the motion capture. The force plate is connected to any free channels on the analog board of the measurement computer, see chapter “How to use analog boards” on page 356.

Kistler force plates must be controlled via a digital I/O cable. For instructions on how to connect the Kistler force plate see chapter “Connecting Kistler force plates” below.

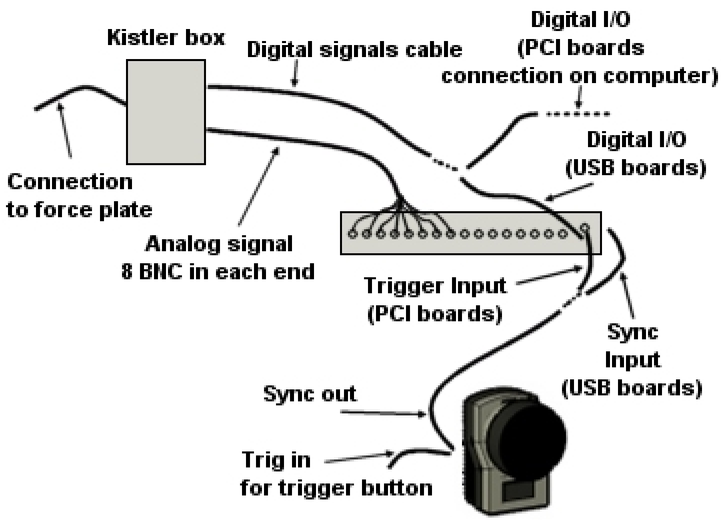
Note: The force plate must have an analog output with an output voltage between ± 10 V. However only the force plates listed in the chapter “Supported force plate devices” on page 363 can be used if you want to calculate the force in QTM.

Note: The channels must be activated on the page for the analog board in the **Project options** dialog, see chapter “Channels” on page 165.

Connecting Kistler force plates

Kistler force plates must have both the analog channels and a control cable connected to the analog board. The following is a description of how to install a Kistler force plate in QTM. It only describes the connection from the Kistler connection box to the analog board. For a description of the connection between the force plate and the connection box, please refer to the Kistler manual.

Start with the hardware connections to the analog board. The picture below is an example of the setup.



1. Make sure that the force plate is working in Kistler's BioWare.
2. Check whether you have a Kistler connection box 5606 or Kistler control unit 5233A.
3. Connect the analog signals (8 BNC cables) from the Kistler box to 8 channels on the analog board.

a. Make sure that the analog channels are connected in the correct order.

b. Do not connect the analog signals from one force plate to different analog boards.

4. Connect the Digital cable from the Digital I/O on the analog board where the Kistler force plate is connected. The force plate is then controlled with the settings on the **Force plate control settings** page in the **Project options** dialog.
The connection differs between the analog boards. Check that you have the correct cable with Qualisys AB.

USB-2533
There are two Digital I/O ports (DSUB15) on the front of the analog board.

5606
Connect the DSUB15 end of the cable (230118) to port A on the analog board. Connect the DSUB37 end of the cable to the Digital input on the 5606 box, i.e. the bottom right connector.

5233A
Connect the DSUB15 end of the cable (230129) to port A on the analog board. Connect the DSUB37 end of the cable to the DSUB37 connector on the 5233A unit.
Remember to press the Remote on the 5233A unit to activate the digital control.

USB1616-FS
There is one Digital I/O port (DSUB15) on the front of the analog board.

5606
Connect the DSUB15 end of the cable (230128) to the analog board. Connect the DSUB37 end of the cable to the Digital input on the 5606 box, i.e. the bottom right connector.

5233A
Connect the DSUB15 end of the cable (230129) to port A on the analog board. Connect the DSUB37 end of the cable to the DSUB37 connector on the 5233A unit.
Remember to press the Remote on the 5233A unit to activate the digital control.

PCI-DAS6402/16
There is one Digital port (DSUB37) on the back of the computer. It is not included automatically with the board.

5606
Connect the cable (230123) between the port on the back of the computer to the Digital input on the 5606 box, i.e. the bottom right connector.

The Digital I/O is also used to reset the force plate before a measurement. It is important to not stand on a Kistler force plate when the reset signal is sent. It is sent at the following operations in QTM:

New file

Changing a setting in Project option in QTM during preview

Just before the Start capture dialog is opened before a capture

In a batch capture just before QTM starts Waiting for next measurement/trigger

5. Remember to check that the sync out signal from the camera system is connected to the external trigger input on the analog board, see chapter “Hardware synchronization” on page E - 13. Otherwise the analog capture will not start.

This is all of the connections that is needed to connect the force plate to the analog board. Then you must add the force plate in QTM follow these steps:

1. Go to **Analog board (...)** page in the **Project options** dialog. Make sure that it is the analog board where the analog channels and the digital I/O cable are connected.
   a. Activate the channels for the force plate. You can rename the analog channels to the Kistler signal names so that it is easier to define the channels for the force plate.
   b. Set the **Sample rate** for the analog board, it is specified in multiples of the marker capture rate. For normal gait measurements you can use a sample rate of 600-1000 Hz. For sport measurements you need a bit higher sample rate.
   c. Go to the **Force plate control settings** page for the analog board and select the number of Kistler force plates that you want to control, also select internal or external amplifier.
   Note: The USB1616 box can only send one control signal per board. However the signal is split in the cable (230128) so that both force plates gets the same control signal. Therefore you can use two force plates, but the plates must use the same ranges.
2. Create the force plates on the **Force data** page in the **Project options** dialog.
3. Open the **Force plate** page.
   a. Enter all of the calibration parameters for the force plates, see chapter “Kistler force plate calibration parameters” on page 193. They are found in the manual for the force plate. Write down the XY and Z range that is used, you need to enter it later for the force-plate control.
   b. Select the correct analog channels for each force-plate, see chapter “Kistler force plate settings” on page 196.
4. Enter the position of the force-plate, see chapter “Force plate location” on page 199. It is good to do this after each new calibration especially if the calibration L is not placed at the same position.
5. Activate the **Calculate force data** option on the **Processing** page. To see the force both in preview and in captured file make sure that it is activated both for **Real-time actions** and **Capture actions**.
6. Test the force plates in QTM. If there is no force, first check that there is no signal on the analog channels.
   a. If there are signals on the analog channels the error is in the settings in QTM. Check the steps 1-5 above.
   b. If there are no analog signals.

   Disconnect the digital I/O cable from the Kistler unit and connect the cable for BioWare.

Start a measurement in BioWare and then start a measurement in QTM at the same time. The analog signals is sent to both BioWare and QTM, so you can see the force data in both.

If there is now analog signals in QTM the control signals are not sent from QTM to the Kistler force plate. Connect the digital I/O cable again. Check that it is connected to the correct ports and that the Remote button is pressed on the 5233A control unit.

Note: It is possible to see the current ranges on the 5233A control unit and on some Kistler amplifiers. Check them so that the ranges are the same as in QTM.

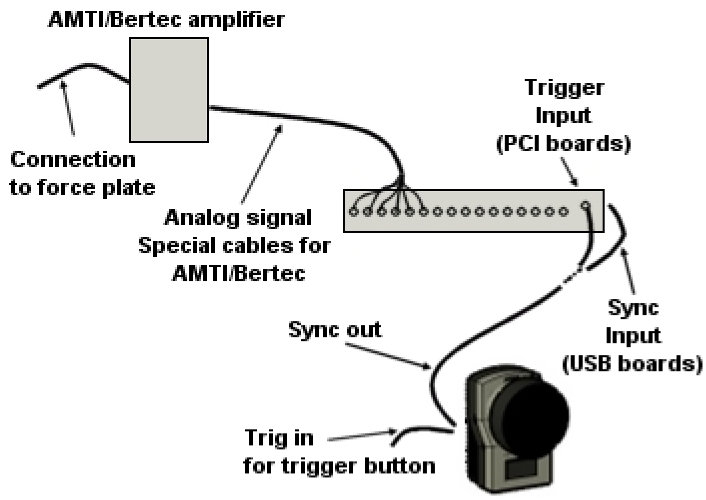
If there is still no analog signal the BNC cables must be connected to the wrong channels. Double-check the connections.

Note: If more than one USB1616FS board is connected to the computer check the order of the boards, see chapter "Installing several USB-1616FS boards" on page E - 10.

Connecting AMTI and Bertec force plates

For AMTI and Bertec force plates only the analog channels needs to be connected to the analog board. The following is a description of how to install a AMTI or Bertec force plate in QTM. It only describes the connection from the AMTI or Bertec amplifier to the analog board. For a description of the connection between the force plate and the amplifier, please refer to the AMTI or Bertec manual.

Start with the hardware connections to the analog board. The picture below is an example of the setup.



1. If you have AMTI's or Bertec's software make sure that the force plate is working in those programs.
2. Connect the analog signals (8 BNC cables) from the AMTI or Bertec amplifier to 8 channels on the analog board.
   a. Make sure that the analog channels are connected in the correct order.
   b. Do not connect the analog signals from one force plate to different analog boards.
3. Remember to check that the sync out signal from the camera system is connected to the external trigger input on the analog board, see chapter "Hardware synchronization" on page E - 13. Otherwise the analog capture will not start.

This is all of the connections that is needed to connect the force plate to the analog board. Then you must add the force plate in QTM follow these steps:

1. Go to **Analog board (...)** page in the **Project options** dialog.
    a. Activate the channels for the force plate. You can rename the analog channels to the force plate signal names so that it is easier to define the channels for the force plate.
    b. Set the **Sample rate** for the analog board, it is specified in multiples of the marker capture rate. For normal gait measurements you can use a sample rate of 600-1000 Hz. For sport measurements you need a bit higher sample rate.
2. Create the force plates on the **Force data** page in the **Project options** dialog.
3. Open the **Force plate** page.
    a. Enter all of the calibration parameters for the force plates, see respectively chapter “AMTI force plate calibration parameters” on page 191 and “Bertec force plate calibration parameters” on page 196.
    Note: AMTI is supplied with the calibration file, use it to import the calibration matrix.
    Note: Bertec usually only supplies the diagonal of the calibration matrix in the manual.
    b. Select the correct analog channels for each force-plate, see respectively chapter “AMTI force plate settings” on page 192 and “Bertec force plate settings” on page 197.
4. Enter the position of the force-plate, see chapter “Force plate location” on page 199. It is good to do this after each new calibration especially if the calibration L is not placed at the same position.
5. Activate the **Calculate force data** option on the **Processing** page. To see the force both in preview and in captured file make sure that it is activated both for **Real-time actions** and **Capture actions**.
6. Test the force plates in QTM. If there is no force, first check that there is no signal on the analog channels.
    a. If there are signals on the analog channels the error is in the settings in QTM. Check the steps 1-5 above.
    b. If there are no analog signals. Check if the BNC cables are connected to the wrong channels.
    Note: If more than one USB1616FS board is connected to the computer check the order of the boards, see chapter “Installing several USB-1616FS boards” on page E - 10.

Supported force plate devices

QTM supports calculation of force data from the force plates of the following manufacturers: AMTI, Bertec and Kistler. For information about how to use the force data see chapter “Force data calculation” on page 311. To calculate the force the settings for the force plate must be specified on the **Force plate** page in the **Project options** dialog, see chapter “Force plate” on page 191.

How to use EMG

EMG devices

With an EMG device the muscle activity can be measured together with the motion capture. QTM can collect EMG data directly from Mega ME6000, Noraxon TeleMyo 2400T G2 and TeleMyo DTS. For instructions on how to connect these EMG devices see “Wireless EMG systems” on page F - 1.

It is also possible to connect an EMG device to an analog board. Then the EMG device is connected to any free channels on the analog board of the measurement computer, see chapter “How to use analog boards” on page 356.

Note: The channels must be activated on the page for the analog board in the **Project options** dialog, see chapter “Channels” on page 165. The EMG device must have an analog output with an output voltage between ± 10 V.

QTM does not support any analysis of the EMG data within the program, i.e. it is only voltage output that is saved. To analyze the data export it to an analysis software.

How to use DV/webcam video devices

Video capture

QTM can record video data, including the sound, together with the motion capture data. This data is only for documentation processes and are not used in the calculation of 3D data.

The video capture starts and stops in synchronization with the motion capture. The synchronization is done via Windows OS, therefore exact synchronization cannot be guaranteed. It is, however, within a quarter of a second. It is possible to change the offset of the video to get a more accurate synchronization, see chapter "Video offset" on next page.

The capture rate of the video data depends on the hardware of the video device, on the settings of the video device and on the general performance of the measurement computer that QTM runs on. To set the settings of the video device, right-click in the view of the video camera in a 2D view window, in the preview mode, to open the **View window** menu. Click **Video camera settings** to open the dialog with settings. For information about the settings see the manual of the video device.

The recorded video data will be saved as an AVI file in the same folder that is selected for the capture file (QTM file) and it will have the same name as the capture file. It can be played in any media player that supports the AVI format as well as viewed in QTM together with the motion capture data. For more information about the display of video data see chapter "DV/webcam video in 2D view" on page 42.

DV/webcam devices

Most DV/webcam devices that support the Microsoft DirectShow API can be used with the QTM software. However the camera must be equipped with a DV–output or FireWire 4-pin connection, to be able to stream out video in real-time. Some low cost DV cameras are equipped with a USB output which cannot be used to stream the video.

There can be several devices connected to QTM, but there might be problems with the video capture if too many are connected. The reason is that the video data stream becomes too large and the frame rate of the video devices will therefore be lowered.

If you have problems capturing many video cameras at the same time you can use import the link to a video file after the measurement, see chapter "Import video link" on next page.

The video devices are connected to QTM on the **Video devices** page in the **Project options** dialog, see chapter "Video devices" on page 169.

If you run ProReflex in **Immediate** mode, the rate of the video capture might be slightly reduced. However, to improve the rate QTM could be set to **Frame buffering** or **Segment buffering** mode.

Note: For information about how to connect the video device to the computer see the manual of the video device.

Compression of video from video cameras

QTM can compress the video recording during the capture of video data. To activate the compression for a video camera, right-click in the view of that camera in a 2D view window. Then click on **Choose video compression** and select the codec you want to use. You have to select the codec you want individually for each video camera that is used in the capture. The recommended codecs are the same as for the Oqus video, see chapter “Codecs for Oqus high-speed video files” on page 261. For DV/webcams the quality is usually good enough with the FFDS-MJPEG option. It is important to set the codec settings according to the description in “FFDS-MJPEG codec settings” on page 154.

The codec settings can be changed by choosing **Video compression settings** on the menu. This will open the settings for the currently selected codec. However it is important to right-click in the correct video view because the codec setting is done individually on each DV/webcam.

Since QTM uses standard Microsoft DirectShow routines for video playback, the video can be compressed after it has been captured as well. In other words, you are free to compress the measurement video file with an external tool such as VirtualDub (www.virtualdub.org) or NanDub.

Note: Qualisys AB does not offer support on video compression issues.

Video offset

Because the Video synchronization is done via Windows it is not always exact. However there is a way to set the offset for your current setup, follow this procedure:

1. Open the **View window** menu by right-clicking in an open Video view window.
2. Then click on **Set time offset** and enter the starting time of the video in the dialog. The starting time is in the measurement time, i.e. with a 1 s starting time the video file will start at 1 s when looking in the 3D view window.
3. Play the file and check that it is ok.
   Note: Because of the offset there will be a part in the beginning or at the end with no video data.
4. Repeat the steps for each video file that has been captured. The offset will be remembered for the video device until it is unplugged from the computer.

Import video link

The import video link feature can be used when a video files have been captured externally from QTM. However the start of the video capture must be triggered in some way to ensure the synchronization with the marker capture. How to trigger the video device can be found in the video manuals, but not all video devices can be triggered externally.

To import the video file click **Import**/**Add link to video file** on the **File** menu. Browse to the video file which must be an AVI-file. The video file do not need to be the same length as the marker capture, since the video length is encoded in the file so that the video frames will be placed at the correct place. If the video has not started in synchronization with the marker capture, an offset can be applied to the video see chapter “Video offset” above.

How to use Point Grey cameras

Point Grey cameras

The Point Grey cameras are designed to stream video data directly to a computer. QTM supports any of the Grasshopper models from Point Grey. However the only the models GRAS-20S4M/C and GRAS-03K2M/C have been tested, if you want to use another model please contact Qualisys AB first. The frame rate and resolution depends on the model, see www.ptgrey.com/products/grasshopper. How to connect and use the Point Grey camera is described in the chapters below.

Computer and other hardware

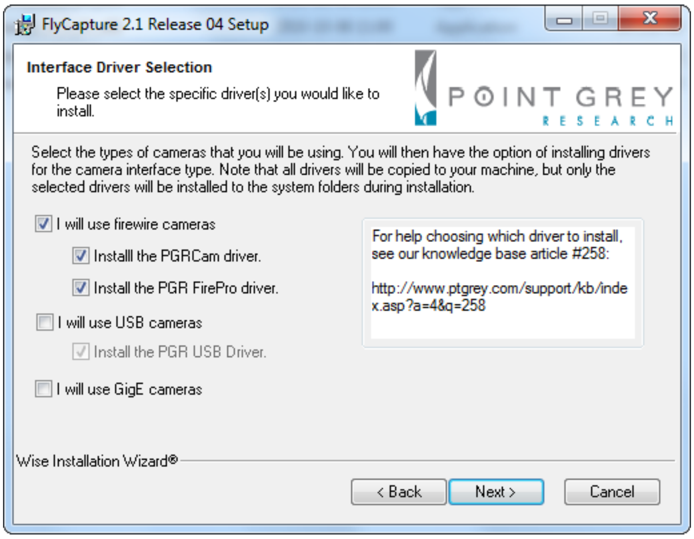
Because the video is streamed directly to the computer it is important that the computer used with the system is fast enough and has a lot of memory. We recommend using Windows 7 64 bit so that you can use more than 4 GByte of memory.

It is also necessary to install a Firewire communication card from Point Grey (1394b PCI express card). It is available in a single a dual bus configuration. If you are going to use two cameras it is important that you use the dual bus version so that the data can be streamed at the maximum frame rate.

Installing the Point Grey camera

The following instructions are for installing Point Grey Grasshopper cameras. It is very important to follow the instructions below exactly for the best performance.

1. Install the Firewire card from Point Grey into a free PCI-express slot in the computer.
2. Check that you have installed QTM 2.5 and that you have installed the Point Grey driver when installing QTM. The driver is called FlyCapture2.dll and is place in the Qualisys Track Manager folder. If the driver is missing install QTM again and make sure that you install the driver in the component selection.
3. Download and run 'http://www.qualisys.com/qualisys/software/flycapture2.1.3.4_x86.exe' (32 bit) or 'http://www.qualisys.com/qualisys/software/flycapture2.1.3.4_x64.exe' (64 bit).
    a. Select the **Complete** installation type.
    b. Select **I will use firewire cameras** for Interface driver selection.



c. Do NOT uncheck the setting for disabling the idle state.

4. Check that the drivers are correct for the standard firewire ports. If you do not change the standard ports they will not work well with other firewire devices.

   a. Start the Windows Device manager.

   b. Go to the entry **PGR IEEE 1394 Bus host controllers**.

   c. If there are any devices with **1394a** in the name. Open the **Properties** and select **Update driver**.

      - Select the alternative **Browse my computer for driver software**.
      - Then select the alternative **Let me pick from list of device drivers on my computer**.
      - Finally select the **1394 OHCI Compliant Host Controller**.

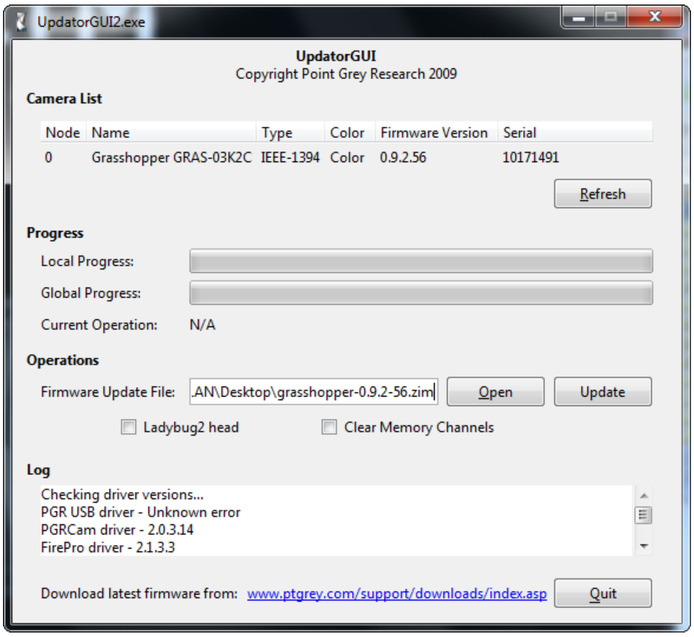
5. Connect the cameras to the Firewire card. You can use any of the firewire ports on the camera and the card. Make sure that you tighten the screws of the connectors.

6. Download the Point grey camera firmware 'http://www.qualisys.com/qualisys/software/grasshopper-0.9.2-56.zip' and unpack it.

   a. Unzip the file and place it somewhere you can find it.

   b. Go to Windows Start menu/Programs/Point Grey Research/Utilities/. Start the program UpdatorGUI2.

   c. For each camera in the list follow the instructions below to update the firmware.



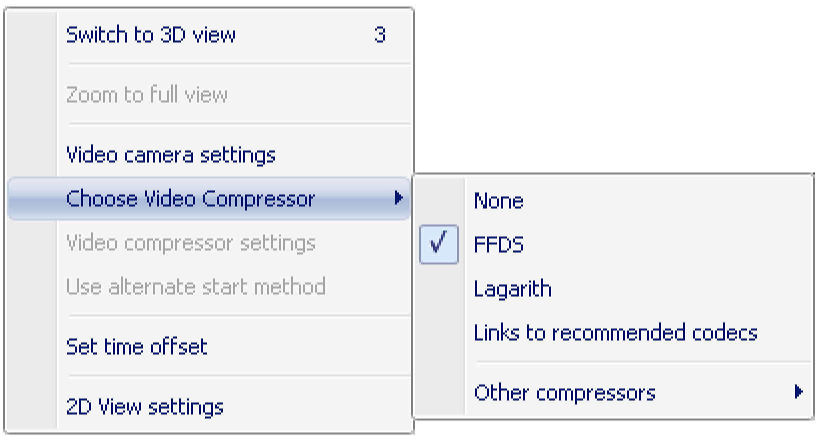
- Select a camera in the list.
- Click on **Open** and browse to the firmware file grasshopper-0.9.2-56.zim.
- Click on Update and wait for the process to finish.
- Repeat the steps for all cameras.

7. Start QTM and verify that you can enable the cameras on the **Video devices** page in the **Project options** dialog and that you can see them in the 2D view window. For instructions how to use and synchronize the cameras see chapters “Point Grey camera settings” on next page and “Synchronization with the Oqus camera” on page 372.

   Note: You can also start FlyCap to see if the cameras work there. However FlyCap sometimes cannot display the camera image even though it works in QTM.

Capturing video with Point Grey

To capture video from the Point Grey camera just activate it on the **Video devices** page in the **Project options** dialog. Then start a new measurement and you will see the Point Grey video in the 2D view window. The settings for the Point grey camera is accessed by right-clicking in the view of the camera.



Click **Video camera settings** to open the **Point Grey camera settings** dialog. This is where you change the frequency and synchronization of the camera, see chapter "Point Grey camera settings" below.

It is recommended that you use the synchronization with the Oqus camera, since otherwise the camera works just like any other video camera. How to synchronize is described in chapter "Synchronization with the Oqus camera" on page 372.

The other important setting is **Choose video compressor**. Because of the amount of data streamed to the computer it is best to compress the video image so that the writing time to the hard drive is minimized. The recommended codec for a fast compression is the MJPEG codec in the ffdshow package. Select **FFDS** and change the settings according to the instructions in the chapter "FFDS-MJPEG codec settings" on page 154.

It is important to notice that the camera changes the exposure time and gain automatically, which means that in a dark room the image will look very noisy. In that case either open the aperture or increase the lighting in the room.

The video is saved in an avi-file that uses the OpenDML 1.02 extension of the AVI standard. This means that the file size is almost unlimited, the standard avi format cannot be larger than 2 Gigabyte.

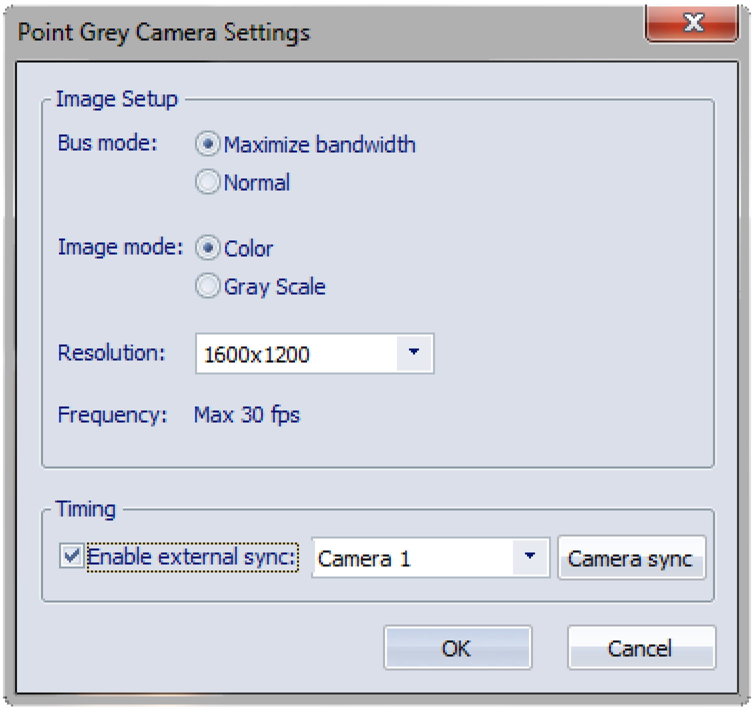
Since the video is streamed to the computer there can be dropped frames during a capture. This is saved as dropped frame in the avi-file, how it is displayed in external programs depends on how it is implemented. In QTM when the frame is dropped the previous frame will be displayed instead, but there is a red exclamation mark in the top left corner. Place the mouse over the red exclamation mark to see the total number of lost frames. If there are too many lost frames lower the frame rate and check that you use the MJPEG codec.

Note: If you use QVA to track a feature in the video the 2D data will be imported correctly so that the lost frames have no 2D data.

Point Grey camera settings

To make the camera more user friendly the settings for the Point grey camera has been limited in QTM compared with the FlyCap. If you need another setting which is available in FlyCap please contact Qualisys AB.

Right-click in the view window of a Point Grey camera and select **Video camera settings** to open the **Point Grey camera settings** dialog.



Bus mode
Select the mode and thereby the speed of the camera.

> **Maximize bandwidth**
> In the **Maximize bandwidth** mode the camera streams the video frames as fast as possible. This is the recommended setting when synchronizing with the Oqus camera. This mode is faster than the **Normal** mode, because the camera is in a mode called Mode 7 and then the images are color interpolated on the computer. The selected color interpolation scheme is called Edge-sensing.
> Note: If the video is not synchronized the video capture will be done at the maximum frequency of the current resolution. This will displayed as **Unknown frequency**, for example at 1600x1200 it will be slightly more than 30 Hz. Because it is not an exact frequency it is strongly recommended to use synchronization with this mode.
>
> **Normal**
> In the **Normal** mode the camera streams the video data in the selected frequency of the **Frequency** setting. This is the recommended setting if you do not synchronize with the Oqus camera. The camera captures the video in the RGB mode, and the available frequencies are the same as in FlyCap for that mode.

Resolution
Select the resolution of the video.

> In the **Maximize bandwidth** mode the maximum frequency that can be used for the synchronization is displayed below the setting. For example for the GRAS-20S4M/C at **1600x1200** the maximum frequency is 30 Hz.
>
> In the **Normal** mode you have more options for the resolution and you can then see the available frequencies on the **Frequency** setting.

Timing
The Timing options are used to set up the synchronization with the Oqus system, for a description of the connection see chapter “Synchronization with the Oqus camera” on next page.

Activate the synchronization with **Enable external sync**. Then select which Oqus camera the Point grey camera is connected to with the drop-down box.

Change the Synchronization output of the selected camera by clicking the **Camera sync** button. It will open the **Advanced Timing** page in the **Project options** dialog, see chapter “Synchronization output” on page 137.

You can use any of the Synchronization output modes, except of course **Measurement time**. However if you will capture markers at different frequencies it is best to use the **Camera independent frequency** mode. Then the Point Grey camera will always use the specified frequency and not change with the Oqus frequency.

The Point grey camera is set to capture on a negative edge, so keep the **TTL signal polarity** setting on **Negative** which is the default for Oqus. The trigger mode that is used in the Point grey camera is called mode 14.

Synchronization with the Oqus camera

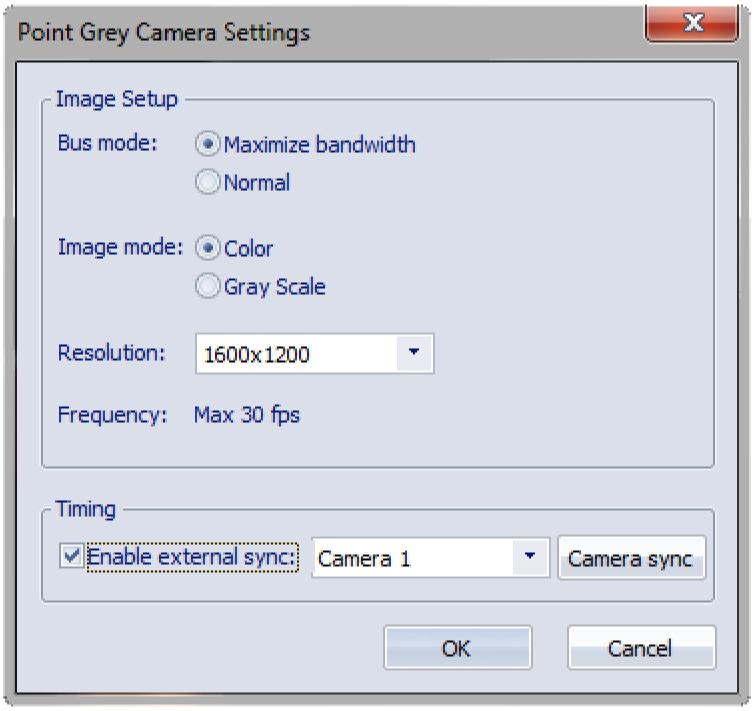
The major advantage with a Point grey camera is that each video frame can be synchronized with the Oqus data. For example if you capture markers at 120 Hz and the Point grey video at 30 Hz. Then the video frames will be captured at every fourth marker frame.

To setup the synchronization you must connect Sync out on the Oqus system to the BNC connector on the sync cable for the Point Grey camera. Connect the Point grey sync cable to the Point grey camera, make sure that you tighten the connector. The sync cable can be connected to any of the Oqus cameras, e.g. you can use separate Oqus cameras sync out signal for two Point Grey cameras or for the analog board and Point grey cameras.

If you have more than one camera the Sync out signal must be connected to the BNC on all of them. The cable to connect the sync signal to the Point grey camera is supplied by Qualisys AB.

Then you must follow this procedure in QTM.

1. Check that the Point Grey camera is activated on the **Video devices** page in the **Project options** dialog.
2. Start a new measurement in QTM.
3. Right-click on the view of the Point Grey camera and select **Video camera settings**.



4. It is recommended to use the **Maximize bandwidth** so that you get the highest possible frame rate.

5. Then select **Enable external sync** select the Oqus camera which the Point Grey camera is connected to.

6. Click on **Camera sync** to open the **Advanced Timing** page in the **Project options** dialog.

Synchronization output	
Mode	Camera independent frequency
Output frequency [1-100000Hz]	100
Duty cycle [%]	50.000
TTL signal polarity	Negative
Output period time [µs]	10000
Pulse duration [µs]	5000

   a. Set the Synchronization output so that you get the correct output frequency. If you always want to use the same frequency for the Point Grey camera it is best to use the **Camera independent frequency** mode.

   b. The Point grey camera is set to capture on a negative edge, so keep the **TTL signal polarity** setting on **Negative** which is the default for Oqus.

7. Close the **Project options** dialog and the **Point Grey camera settings** dialog.

8. The Point Grey camera is now frame synchronized and you can start making captures.

Known issues with Point Grey cameras

There can be dropped frames when capturing the Point Grey video, especially when the camera is using the maximum frequency. It helps to have fast CPU and hard drives and a lot of memory in the computer. If you do get lost frames the synchronization in QTM is still OK. However if you think there are too many lost frames then the solution is to lower the video frequency and disable RT preview in QTM and any background services that may be running on the computer.
Instead of the dropped frames QTM displays the previous video frame, but there is a red

exclamation mark in the top left corner. Place the mouse over the red exclamation mark to see the total number of lost frames.

There is also a known problem with the PCI bus in Windows 7. It causes some images to be distorted, we are waiting for Microsoft to fix this error. Usually it can only be seen now and then, but if it appears often the solution is to lower the frequency.

The standard cable supplied with the Point Grey camera is 4.5 m. If you need a longer cable, there are 10 m cables and repeaters available. Please ask Qualisys AB for more information.

It is not possible to use the pretrigger option with the Point Grey cameras.

Timing hardware

How to use external trigger

An external trigger can be used to trigger the start of the motion capture, see chapter “External trigger (basic)” on page 130. If the movement that is captured is very short the external trigger can be used together with pretrigger so that the whole movement is captured.

For Oqus the trigger is connected to a splitter cable on any Oqus camera in the system. The delay between the trigger and capture start can be set for Oqus, however the minimum is 20 ms. Therefore it is important to use the Sync out signal when synchronizing with other equipment.

On the Oqus camera there can be errors if you send a trigger signal when the camera is setting up the measurement. I.e before it says **Waiting for trigger** in the **Status** bar. The problem can be solved by moving the trigger button to the master camera.

For ProReflex the trigger is connected to the **External trig input** pin of the **Control** port on the master camera, see chapter “Control port” on page B - 12.

Possible external trigger devices are for example:

- Trigger button
- Photo cell
- Other systems

How to use pretrigger

The pretrigger is used to collect frames before the arrival of a trigger event. The pretrigger frames are saved in the camera buffer before the trigger event and after the event the frames are sent to the measurement computer. When using pretrigger it is impossible to know the delay between the trigger signal and the first frame after the pretrigger frames. This is because the cameras are already measuring and the trigger event can come at any time in reference to the frame rate.

Note: For ProReflex it is important to check that all of the pretrigger data fits in the camera buffer.

Note: Pretrigger cannot be used with segment buffering on the ProReflex.

The trigger event can be sent to the camera either with an external trigger or from a dialog in QTM if there is no external trigger. On ProReflex extra hardware must be used with an external trigger to connect the signals, see below.

When using analog boards with pretrigger the connection differs between the boards, see below.

The pretrigger settings can be found on the **Timing** page in the **Project options** dialog, see chapter “Pretrigger (basic)” on page 131.

Measurement with analog capture while using pretrigger

When analog data is collected while using pretrigger, an external trigger must be used to start the capture. Activate the external trigger setting on the **Timing** page in the

Project options dialog. The external trigger is connected to different connectors on the analog board depending on the analog board type and camera type.

USB-2533
It is important that the pretrigger buffer is full before you press the trigger button. Otherwise you will get an error message from the analog board.

Oqus
Connect the **Sync out** signal from the camera to **External trigger** input on the analog board. Then split the **Trig input** signal to the camera with T-connector and connect one end to the trigger button. The other end must be connected to the first analog channel on the board.

ProReflex
Connect the pretrigger boxes to the cameras. Connect the male BNC connector on the trig cable to the **External trigger** input on the analog board. Then connect a T-connector to the female BNC connector on the trig cable. Connect the trigger button to one end and connect the other end to the first analog channel on the board.

USB-1616FS
Does not support pretrigger functionality.

PCI-DAS6402/16
For PCI-DAS6402/16 a jumper must be connected between pin 45 and pin 47 on the A/D board so that the analog board can start the pre-measurement. If a BNC16SE connection box is used with the analog board, a jumper between pin 11 and pin 13 on the P1 connector on the front of the box can be used instead.

The RT/preview mode will not be synchronized because the camera's external trigger signal is used to start the analog capture instead of the normal **Sync out** signal.

Oqus
Split the **Trig input** signal of the camera with a T-connector and connect one end to the trigger button. The other end must be connected to the MCU Sync port on the analog board. This is the BNC connector furthest to the right on the BNC connection box.

ProReflex
Connect the pretrigger boxes to the cameras. Then connect a T-connector to the female BNC connector on the trig cable. Connect one end to the trigger button and the other end must be connected to the MCU Sync port on the analog board. This is the BNC connector furthest to the right on the BNC connection box.

PCI-DAS1602/16

The RT/preview mode will not be synchronized because the camera's external trigger signal is used to start the analog capture instead of the normal **Sync out** signal.

Oqus
Split the **Trig input** signal of the camera with a T-connector and connect one end to the trigger button. The other end must be connected to the MCU Sync port on the analog board. This is the BNC connector furthest to the right on the BNC connection box.

ProReflex
Connect the pretrigger boxes to the cameras. Then connect a T-connector to the female BNC connector on the trig cable. Connect one end to the trigger

button and the other end must be connected to the MCU Sync port on the analog board. This is the BNC connector furthest to the right on the BNC connection box.

Pretrigger hardware when using external trigger (ProReflex)

When collecting pretrigger frames while an external trigger device is used to start the capture, the following additional hardware must be installed:

- One Master unit that is connected to the master camera, i.e. the camera that is connected to the measurement computer.
- Slave units connected to every other camera in the motion capture system.

The pretrigger hardware is connected to the **External sync input** pin of the **Control** port on each camera, see chapter “Control port” on page B - 12. The Master unit and the Slave units are connected by coaxial cables using BNC T-connectors and a 50 ohm resistor at the last Slave unit. The external trigger device is connected to the Master unit of the pretrigger.

How to use external timebase

The external timebase is used to get a required frame rate in the motion capture from an external device. It can also be used to get a periodic frame rate.

Note:The external timebase for ProReflex is only active during the capture, in preview the master camera controls the frame rate.

The settings for external timebase are on the **Advanced** page in the **Project options** dialog, see chapter “External timebase” on page 135.

External timebase connection

For Oqus the external timebase is connected to the sync in connector. Contact Qualisys AB to get a special kind of splitter cable with this connector.

Note: If an external trigger is used to start the measurement, it must be connected to the same camera as the external timebase.

For ProReflex the external timebase signal is connected to the **External sync input** pin of the **Control** port on the master camera, see chapter “Control port” on page B - 12.

External timebase signal with constant period (cycle) time

For an Oqus camera is recommended to use the Periodic mode, see chapter “Using External timebase for synchronization to a periodic signal” on page 382. To use an external timebase signal with constant period time and trigger each frame with a pulse follow the steps below:

1. Set the capture rate in QTM to the same as the period time of the external timebase source.
2. Select **Use external timebase** on the **Advanced** page in the **Project options** dialog. External timebase can be used together with an external trigger. If both are used, the first frame will be captured on the first timebase pulse following the trigger event of the external trigger.
    a. Note: When using external timebase on an Oqus system in the **Non-periodic** mode it is important to take in account the delay between a

sync in signal and the frame capture. The minimum delay is 19 ms. How to compensate for the delay depends on the equipment that you are synchronizing with. For periodic sync signal one way is to set the delay to a number that is A x period time, where is an integer. Then every picture will still be captured in synchronization with the external source. However the frequency cannot be higher than 120 Hz.

3. Use the default setting (10 s) for the **Max expected time between two frames** option. If no frame has been received by QTM within this time period, QTM assumes that an error has occurred and aborts the measurement.
4. Start a new measurement. The frequency will be reported as **EXT** in the main status bar, because it is unknown.
5. Enter the expected **Capture period** in the **Start capture** dialog, decimals can be used.

Warning: DO NOT overclock the cameras by setting the external timebase rate higher than the camera specification. For Oqus the maximum frame rate at full FOV are the following.

Oqus 1-series - 247 Hz
Oqus 3-series - 500 Hz
Oqus 5-series - 180 Hz

It is also important to check that the exposure is not longer than 1/10 of the period time of the external timebase frequency.

For ProReflex MCUs the maximum frame rates are the following:

MCU120 - Maximum frame rate 120Hz
MCU240 - Maximum frame rate 240Hz
MCU500 - Maximum frame rate 500Hz
MCU1000 - Maximum frame rate 1000Hz

The camera will be damaged if it is overclocked and the guarantee is then void.

External timebase with bursts of signals with constant period (cycle) time and with delays between the bursts

To use an external timebase with bursts of signals with constant period time and with delays between the bursts follow the steps below:

1. First the possible time periods of the measurement system must be calculated.
   - Calculate the smallest time interval *t1* between two frames that can occur during the measurement (i.e. the smallest time interval between two pulses from the external timebase source that can occur during the measurement).
   - Calculate the longest interval between two pulses from the external timebase source that can occur during the measurement, multiply it by 1.5 and call it *t2*.
   - Calculate the maximum frequency *f* as 1/ *t1*.
     Note: On the Oqus cameras the non-periodic sync cannot be faster than 120 Hz. This is because of network delays of the TCP/IP network.
   - Calculate the number of pulses *n* that will be generated during the measurement (i.e. the number of frames that will be captured).
2. Set the **Marker capture frequency** in QTM to *f*.
   If the **Capture rate** is set to a value that is lower than *f* the cameras will flash

for a larger part of the time between two pulses, which might lead to overheating of the cameras. It might also result in that the time to calculate marker positions in each frame is too short. Therefore some markers, those vertically lowest, might be lost.

3. Select **External timebase** (and **Non-periodic** for Oqus) on the **Advanced** page in the **Project options** dialog.
   External timebase can be used together with an external trigger. If both are used, the first frame will be captured on the first timebase pulse following the trigger event of the external trigger.
4. Set the **Max expected time between two frames** option to *t2* seconds. If no frame has been received by QTM within *t2* seconds, QTM assumes that an error has occurred and aborts the measurement.
5. Start a new measurement. The frequency will be reported as **EXT** in the main status bar, because it is unknown.
6. Enter the number of frames (*n*) under the **Capture period** heading in the **Start capture** dialog.
   It is important to specify the correct number of frames, so that the last part of the measurement is not excluded. However they should not be too many, if the external timebase source stops firing frames after the measurement that would mean that the measurement is never finished. The best way is to specify a few frames extra and then let the external timebase source fire a little longer than necessary, so that all of the required number of frames will definitely receive a pulse from the external timebase source.

Warning: DO NOT overclock the cameras by setting the external timebase rate higher than the camera specification. For Oqus the maximum frame rates at full FOV are the following.

Oqus 1-series - 247 Hz
Oqus 3-series - 500 Hz
Oqus 5-series - 180 Hz

It is also important to check that the exposure is not longer than 1/10 of the period time of the external timebase frequency.

For ProReflex MCUs the maximum frame rates are the following:

MCU120 - Maximum frame rate 120Hz
MCU240 - Maximum frame rate 240Hz
MCU500 - Maximum frame rate 500Hz
MCU1000 - Maximum frame rate 1000Hz

The camera will be damaged if it is overclocked and the guarantee is then void.

Note: The tracking can become significantly of poorer quality if you use a timebase with a large difference between the smallest and the largest delay between two frames. Since the tracking assumes a constant time period between two frames.

Synchronizing external hardware

How to synchronize external hardware

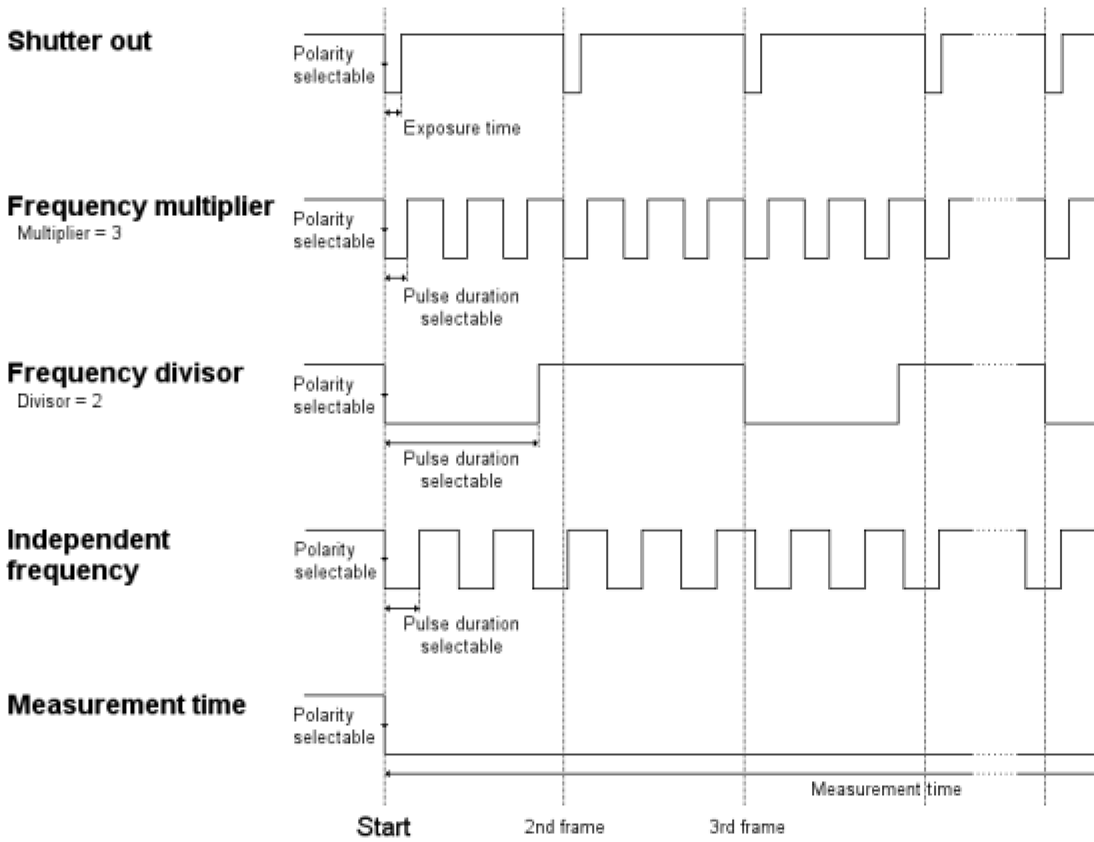
It is possible to synchronize other external hardware that are not directly integrated in QTM. This can be achieved either by triggering the external hardware from the cameras or sending a signal from the external hardware to the cameras. Which option to use depends on the specification of the external hardware, read the following chapters and contact Qualisys AB if you have any questions.

The external hardware must be able to trigger on a TTL pulse or send out TTL pulse according to one of the following alternatives.

Using Sync out for synchronization

The recommended option is to use the **Sync out** signal from the cameras. This is the same signal that is used for synchronizing the analog boards. The **Sync out** signal is a pulse train with a TTL pulse that is only active when the camera is measuring.

On the Oqus camera the signal can be modified to give you different signals see chapter “Synchronization output” on page 137, but the default is to send a pulse for each camera frame where the pulse has the same length as the exposure time. The **Sync out** signal can also be set independently for each camera in the system. Which means that you can set different outputs for different cameras and adapt the output to different hardware. The different outputs of the camera are shown in the image below.



For ProReflex the **Sync out** signal cannot be modified and there is always one pulse for each camera frame see chapter “Control port” on page B - 12 for the definition.

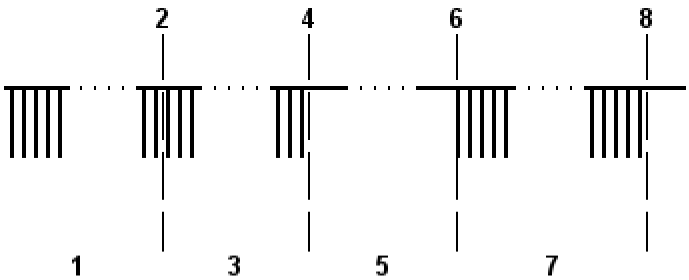
The signal can be used in two different ways:

> **Start the external capture on the first TTL pulse and use an internal frequency**
> In this mode there will be a small drift, between the external hardware and the camera system. How much it will be depends among other things on the frequencies of the two systems and also the accuracy of the external equipment.
>
> The external capture can also be stopped by the camera system if the **Frequency multiplier** mode is used and the pulse duration is the same as the period time. Then the signal will for example be low during the whole measurement until it stops, see chapter “Synchronization output” on page 137.

Frame synchronize the capture of the external hardware
In this mode there is no drift between the two system because the **Sync out** signal is synchronized with the camera frequency. For more information on how to get different frequencies see chapter “Synchronization output” on page 137.

Note: If you want to use a sync output signal that is faster than 5000 Hz on the Oqus camera, then it is recommended that you use the connector on the Master camera. The master camera is displayed with an M next to the ID in the camera list to the left of the settings.

In a measurement the pulse train for the default mode with the setting **Shutter out** will look as the figure below. Which is also the output for the ProReflex camera.

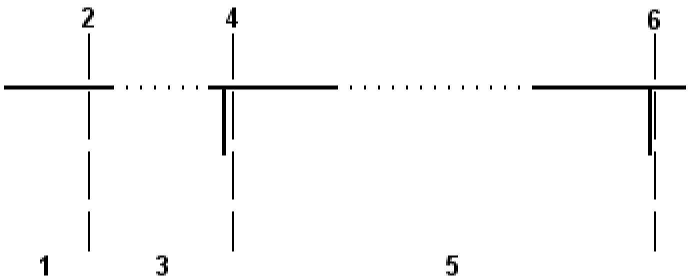


1. **Preview**
   During preview a pulse is sent on each preview frame.
2. **Capture**
   Click on **Capture** in the **Capture** menu.
3. **Start capture dialog open**
   When the **Start capture** dialog is open the pulse continues because the preview is still being updated.
4. **Start**
   Click on **Start** in the **Start capture** dialog.
5. **Waiting for measurement**
   When the camera waits for the start of the measurement the sync output signal is stopped. How long this period is depends mostly on two things.
   a. With external trigger this period continues until you press the button. Therefore we recommend that you use external trigger so that you have time to initialise the measurement on the external device.
   b. Without external trigger the period is less than a second.
6. **Measurement start**
   For the default Oqus setting the measurement starts on the negative edge of the first pulse, which means that the external hardware must trigger on the same edge. For ProReflex, however, the measurement starts on the positive edge.
7. **Measurement**
   During the whole measurement a TTL pulse is sent for each frame.
8. **Measurement stop**
   The measurement stops and the **Sync out** signal is high until the next preview.

   Note: In batch capture there is a couple of preview frames between each measurement. These preview frames will be seen as pulses on the **Sync out** signal and must be considered by the external hardware.

Using Trig in for synchronization

Another way to synchronize is to use the **External trigger** input signal. This signal must be sent to the camera either by a button or by another hardware. However with this method there is a longer delay from the trigger event to the start of the measurement. The signal that is used will look as the figure below.



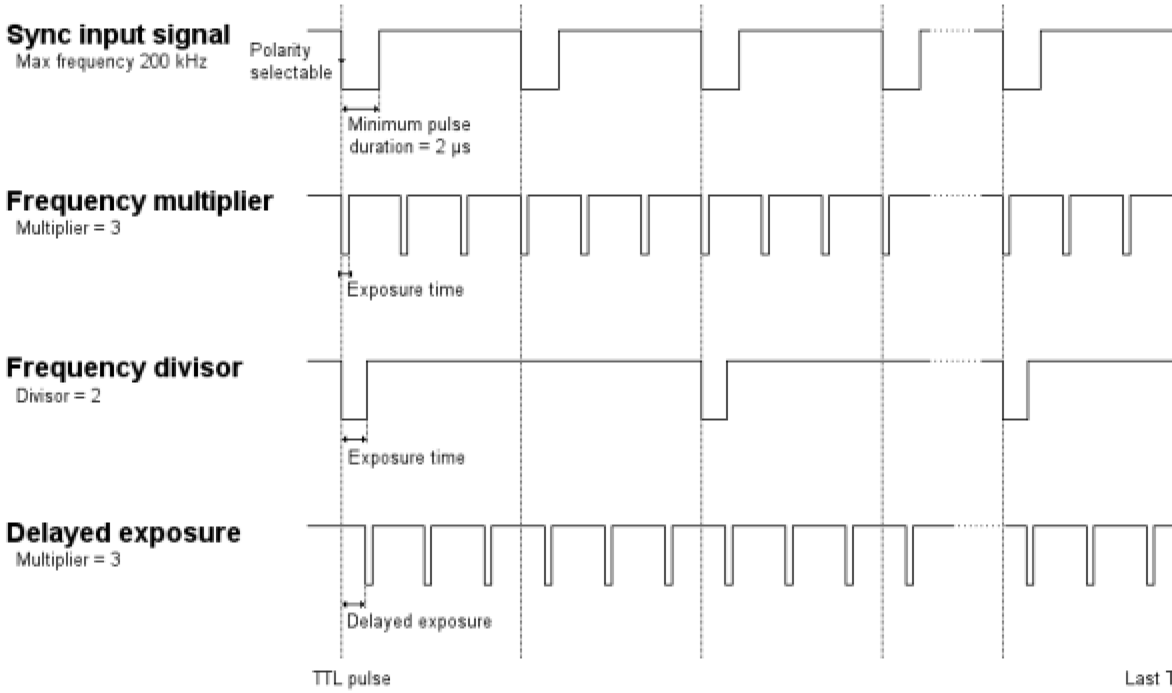
1. **Start capture dialog open**
2. **Start**
   Click on start in the **Start capture** dialog.
3. **Waiting for measurement**
   The camera waits for the trigger event on the **External trigger** signal.
4. **Measurement starts**
   The measurement starts with a short delay compared to the trigger event. For Oqus the delay is adjustable, but the minimum delay is 20 milliseconds and for ProReflex the delay is always 550 microseconds.
5. **Measurement**
6. **Measurement stop**
   The measurement can end in three different ways. Only one of them will have a trigger signal on the **External trigger** input signal.
   a. The measurement comes to the end of the specified measurement time.
      In this case there is no pulse on the **External trigger** signal. The external hardware measurement must be setup to measure for the same time as QTM.
   b. The measurement is stopped manually in QTM.
      In this case there is no pulse on the **External trigger** signal. The external hardware measurement must be stopped manually as well, the stop will therefore not be synchronized.
   c. The measurement is stopped with external trigger.
      In this case there is a pulse on the **External trigger** signal and the measurement will stop on the next frame. Because the stop pulse can come at any time it is impossible to tell the delay until the measurement stops.

Using External timebase for synchronization to a periodic signal

The Oqus cameras has a synchronization input that can be used to synchronize each frame of the camera capture. This means that the camera frequency will depend on the input signal so that there is no drift between the two systems.

The **External timebase** can either trigger each frame of the camera or that the camera locks on to a periodic **Sync in** signal. The recommended option is to lock on to the frequency and then multiply or divide to get the camera frequency. This is because when triggering each frame with the **Edge triggered** option there is a minimum delay of 19 ms from the sync pulse to the camera frame. The following description will only describe the lock on alternative.

The **Sync in** signal from the external hardware must be a TTL pulse and can be divided or multiplied to get the camera frequency that you need, for **External timebase** settings see chapter “External timebase” on page 135.



The **Sync in** signal can be connected to any of the cameras in the system. However it is important that the **Sync in** signal is sent to the camera all the time so that the synchronization is not lost.

Follow these instructions to use external timebase with a periodic signal.

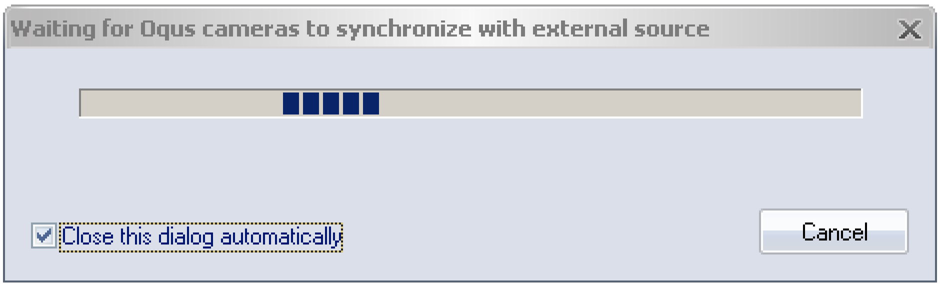
Real-time

1. Make sure that the settings for the **External timebase** on the **Advanced (Timing)** page in the **Project options** dialog are correct for your setup.

   Warning: Do not use **External timebase** settings that results in a camera frequency higher than the camera can use at the full image size.

   Oqus 1-series - 247 Hz
   Oqus 3-series - 500 Hz
   Oqus 5-series - 180 Hz

2. First time the periodic sync is activated you must wait for the camera to synchronize after clicking **New** on the **File** menu. QTM will report **EXT ? Hz** in the status bar for the camera frequency and the dialog below will appear. The following times a measurement is started the camera system will already be locked on and the real-time will start immediately.



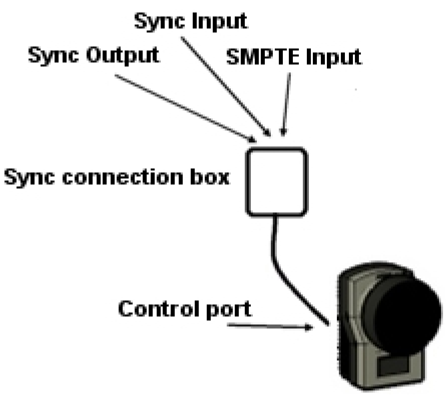
3. When the camera is synchronized the current camera frequency will be reported in the right corner of the status bar. The frequency will also be fixed for the camera system in the **Camera settings** sidebar and in the **Project options** dialog, so that the exposure time settings are limited to the correct values.
4. The synchronized samples can now be sent with the RT protocol to another collection system.

Measurement

5. The start of the capture needs to be known if there is an external system that collects data independently of QTM. There are two ways to know the time of the start.
   a. Record the **Sync out** signal from the camera on the external system. If you use external trigger on both of the systems then the start of the capture is synchronized with the first pulse in the pulse train after the external trigger signal is sent. Otherwise if the camera system is started without a trigger the start of the capture is after a very short pause in the pulse train between when you click on **Start** in the **Start capture** dialog and when the camera starts capturing.
   b. Start the camera system and the external system with the external trigger signal. Then the start of the camera system is in the default mode delayed 20 ms from the trigger pulse, which means that the camera system starts on following Sync in pulse. 20 ms is the minimum delay but the delay can be set to a higher number on the **Advanced (Timing)** page.

Using synchronization box for SMPTE

The Oqus system can also use the SMPTE signal to timestamp and synchronize the measurements. To get the correct signals a special synchronization box must be attached to the camera.



The synchronization box has the following connections.

Sync output
This is the standard Sync out connection from the camera, see chapter “Synchronization output” on page 137 and “Using Sync out for synchronization” on page 380.

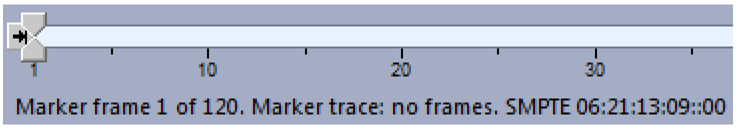
Sync input
This is the Sync in connection for an external timebase connection “External timebase” on page 135 and “Using External timebase for synchronization to a periodic signal” on page 382. For example the world clock of 48 or 96 kHz from a sound sampling board can be connected to this connection and then divided to the wanted frequency on the **Advanced Timing** page in the **Project options** dialog. Make sure that you have set a divisor that gives a camera frequency that works with the current camera resolution.

Note: If you use SMPTE for timestamp and another external signal for synchronization, both must be connected to the same camera.

SMPTE input
This connection is used to convert the SMPTE signal to a signal compatible with the Oqus camera. The SMPTE signal can be used for both timestamp and synchronization.

Note: A trigger button cannot be connected to the camera with the SMPTE signal, because it uses the Trig in pin of the control port.

Timestamp
To activate the SMPTE timestamp open the **Advanced Timing** page in the Project options dialog. Then select **Use SMPTE timestamp** below the **SMPTE** heading, see chapter “SMPTE” on page 139.



The timestamp is displayed in the **Timeline control** bar. Because the timestamp is sampled at 30 Hz there can be several frames per timestamp. Which is indicated by the last number in the timestamp.

Synchronization
To activate the SMPTE synchronization open the **Advanced Timing** page in the Project options dialog. Then select **SMPTE** as **Signal source** below the **SMPTE** heading, see chapter “External timebase” on page 135.

Troubleshooting QTM

Troubleshooting connection

Symptoms

QTM cannot find the connection to the camera system or do not find the whole system.

Resolutions

- Check that the camera system is turned on.
- Check that the cable between the computer and the master camera is connected.
  Note: For ProReflex, on laptops check that the cable to the PC-card is connected.
- Check that the cables between the cameras in the camera system are connected.
- Check that the cameras are working and that the cables are not damaged.
  Note: Do not remove more than one component at a time when looking for the error.
- Check that the connection settings on the **Connection** page in the **Project options** dialog are OK. Check especially if the **Advanced** option has been used.
- For Oqus, check that the network interface has the correct settings, see “Network card setup” on page 341.
- For wireless Oqus systems, check that the computer is connected to the correct wireless network (SSID).
- Restart the system in the following order: exit QTM, shut down the camera system, turn on the camera system, wait until all cameras are powered up and then start QTM.

Symptoms (Oqus)

The Oqus camera startup stops at 3/4 of the status bar. This means that the camera has not got an IP-address.

Resolutions

- Restart the computer.
- Check that QDS is running.
- Check that the cable between the computer and the master camera is connected.

- If the computer has two network cards remove the other network cable and go to Network connections in Windows and check that the correct network is connected.
- Check that the network has the correct static IP-address settings, see “Network card setup” on page 341. Run the QDS wizard if you do not know how to change the IP-address settings.
- For wireless Oqus systems, check that the computer is connected to the correct wireless network (SSID).
- For wireless Oqus systems, check that the wireless Oqus camera has a small antenna in the display. If not connect the system to a wired network and run the QDS wizard, see “Oqus wireless camera setup wizard” on page 335.
- Check that QDS is not blocked by a firewall. Most importantly check that the Windows firewall allows the program.
  Note: On Windows XP the exceptions in the Windows firewall can change if you disconnect another network card than the one you use for the Oqus cameras. In that case restart QDS to get a new message from the firewall.
- Make sure that you are using the built-in wired Ethernet port. QDS does not work with some Ethernet adapters.

Troubleshooting calibration

Symptoms

The calibration is inexact, i.e. too few points or large residuals.

Resolutions

- Check that the **Exact wand length** on the **Calibration** page in the **Project options** dialog are correct.
- Check that all cameras can see the L-shaped reference structure.
  Note: This does not apply to an extended calibration.
- Check that the calibration wand has not been moved too fast, since the frame rate is only 50 Hz during the calibration.
- Check that the calibration wand has been moved inside the calibration volume during the calibration, i.e. check that there is at least 500 points used for each camera in the **Calibration results** dialog.
  Note: If the wand is larger than the calibration volume, change to a smaller wand calibration kit.
- Check that the camera positioning is OK, see chapter "Camera positioning" on page 250.

Symptoms

The traces of the calibration wand trajectories are cut in a plane and are segmented.

Resolution

- Check that the **Bounding box** setting on the **3D Tracking** page in the **Project options** dialog are large enough.

Symptoms

The field of view of the cameras is too small.

Resolution

- Move the cameras out from the measurement volume.

Symptoms

The camera result of a camera says **Unused camera**.

Resolution

- The camera is deactivated on the **Linearization** page in the **Project options** dialog. Activate the camera and perform the calibration again or reprocess the calibration, see chapter "Reprocess calibration" on page 244.

Troubleshooting capture

Symptoms

There are no visible markers in the **2D view** window in the preview mode.

Resolutions

- Check that the cameras are focused. Because of the size of the sensor this is more important for Oqus 3- and 5- series than for Oqus 1-serie and ProReflex.
- Check that the aperture is open enough.
- For Oqus, check the **Exposure time** and **Marker threshold** settings, see chapter “Tips on marker settings in QTM” on page 250.

Symptoms

The markers in the 2D view are too small (check the size in the **Data info** window).

Resolutions

- Check that the aperture is open enough.
- Move the camera system closer to the markers.
- Change to a larger marker on the measurement object.
- Check that the marker is not worn, if it is replace it with a new one.

Symptoms (Oqus)

The markers in 2D are not stable.

Resolutions

- Check the **Exposure time** and **Marker threshold** settings, see chapter “Tips on marker settings in QTM” on page 250.

Symptoms (Oqus)

The markers are visible in the preview mode but disappears while capturing at high frequencies (above 500 Hz).

Resolutions

- The camera does not have time to calculate the markers during the measurement because the background is too bright. Either reduce the image size to give the camera less data to process or increase the **Marker threshold** setting to reduce the influence of the background.

Symptoms

The intensity of the markers in the 2D view is too low.

Resolution

- Check the aperture, see chapter “Setting the aperture and focus” on page 343.

Symptoms (Oqus)

There are flickering pixels in the video image.

Resolutions

- Run the installer for the current firmware again.
  Hint: If you want to save time click **Advanced** in QFI and uncheck the

Upgrade firmware option. Because it is only the sensor that needs to be recalibrated.

Symptoms

There are fewer markers in the 3D view than in the 2D view.

Resolution

- Check that the **Bounding box** setting on the **3D Tracking** page in the **Project options** dialog are large enough.

Symptoms

Some of the markers are missing in each frame.

Resolution

- Check that the **Max number of markers per frame** setting is not to low.

Symptoms (Oqus)

QTM is showing camera out of sync in the status bar.

Resolutions

- This usually happens directly after startup when the cameras are being synchronized to each other. Wait until the message disappears. It helps to shutdown preview for a while, because the cameras will have more time to synchronize if they are not measuring, especially if they are running close to their maximum frequency.

Symptoms

QTM is waiting for external trigger, but no trigger is connected.

Resolution

- Check that the **Use external trigger** option is not selected on the **Timing** page in the **Project options** dialog.

Symptoms

A camera is indicated as **Not use for tracking** in the **2D view** and the **3D view** window.

Resolution

- The camera is deactivated on the **Linearization** page in the **Project options** dialog. Activate the camera and retrack the measurement if you have already made measurements, see chapter “Reprocessing a file” on page 281.
  Note: If the camera was deactivated during the calibration, the calibration must be reprocessed if you want to use the camera, see chapter “Reprocess calibration” on page 244.

Symptoms

A dialog which says that the camera system has not been calibrated is shown, even though it has been calibrated.

Resolution

- None or just one camera is activated on the **Linearization** page in the **Project options** dialog. Activate the cameras.

Symptoms (Oqus)

There are error messages saying that the camera did not reply three times in a row.

Resolution

- Check that you are using the built-in wired Ethernet. Sometimes the communication does not work with Ethernet adapters.
- Check the Ethernet cables so that they are not broken.

Troubleshooting tracking

Symptoms

There is too much segmentation in the capture file.

Resolutions

- Check that the **Maximum number of markers per frame** setting is not to low.
- Check the tracking parameters, see chapter “3D Tracker parameters” on page 172.
- Make a new calibration.
- Check that the camera positioning is OK, see chapter “Camera positioning” on page 250.
- Check that the calibration wand has been moved in the area where the segmentation occurs. E.g. when segmentation occurs close to the floor at gait measurements.

Symptoms

The trajectories are swapping.

Resolutions

- Check if the markers on the measurement object can be smaller.
- Check if the markers on the measurement object are too close to each other. Move them further apart to reduce swapping.
- Lower the **Max frame gap** setting on the **3D Tracking** page in the **Project options** dialog. A large **Max frame gap** will make it more likely that some trajectories will swap.
- Check the tracking parameters, see chapter “3D Tracker parameters” on page 172.

Symptoms

There are large gaps between two 3D points in trajectories.

Resolutions

- Check that the frequency is high enough.

Symptoms

There are more trajectories than markers.

Resolution

- Increase the **Max frame gap** setting on the **3D tracking** page in the **Project options** dialog. It can be increased up to 127 frames. However with a large **Max frame gap** the more likely it is that some trajectories will swap.

Symptoms

AIM cannot find a solution or finds the wrong solution.

Resolution

- Make sure that it is the correct AIM model that is applied.
- Make sure that the model was generated from a file where all of the trajectories could be viewed during the whole measurement and where there were no erratic data, see chapter “Generating an AIM model” on page 302.

Symptoms

The markers in the **3D view** window are located in one plane.

Resolution

- 2D tracking has been used instead of 3D tracking. If you want 3D tracking in the following captures change to **3D** on the **Processing** page in the **Project options** dialog. To 3D track a saved capture file with 2D tracked data, you have to Batch process the file with 3D tracking, see chapter “Batch processing” on page 279.

Troubleshooting reflections

Symptoms

The cameras capture unwanted reflections.

Note: You can use the **Video**/**Viewfinder** mode to find the reflections.

Resolutions

- Check that there are no other sources of infrared light in the measurement volume, e.g. reflections of the sun, lamps etc.
- Check that the cameras cannot see each other IR flashes or the reflection of the IR flashes.
- Check that there are no very glossy materials in the measurement volume.
- For Oqus, use the marker masking function if you cannot remove the reflections, see chapter “How to use marker masking” on page 264 and “How to use auto marker masking” on page 265.

Troubleshooting force calculation

Symptoms

The forces are too high or too low.

Resolution

- Check that the calibration parameters for the force plate have been entered correctly, for information about the settings see chapter "Force plate" on page 191.

Symptoms

QTM does not calculate the force data.

Resolution

- Check that the **Calculate force** option have been selected for the force plate on the **Force data** page in the **Project options** dialog.

Symptoms

The force vector is placed at the center of the force plate in the **3D view** window.

Resolution

- Check the **COP threshold** on the **Force plate** page in the **Project options** dialog. If the threshold is too high the COP is not calculated and then the vector is placed at the center.

Symptoms

The force vector is pointing in the wrong direction.

Resolution

- Check the settings for the force plate location on the **Force plate** page in the **Project options** dialog.

Symptoms

The purple square representing the force plate is not placed at the correct place in the **3D view** window.

Resolution

- Change the **Force plate location** settings on the **Force plate** page in the **Project options** dialog.

Troubleshooting 6DOF

Symptoms

There is no 6DOF body in the **3D view** window.

Resolution

- Check that **Calculate 6DOF** is activated on the **Processing** page in the **Project options** dialog. If you have a file reprocess it with **Calculate 6DOF** activated with the correct 6DOF bodies.
- Check that the 6DOF body is within the measurement volume.
- Check that the 6DOF body definition includes at least four points, which are not in one plane.
- Check that at least four of the markers on the 6DOF body can be captured at the beginning of the measurement.
- Check that at least three of the markers can be seen in all frames during the measurement. Otherwise the 6DOF tracking function cannot find the body in those frames.

Symptoms

There is no 6DOF body data in the **Data info** window even though it is displayed in the **3D view** window.

Resolution

- Check if the 6DOF body uses the **Use the current position of this rigid body** option in the **Coordinate system for rigid body data** dialog. If it does it will not be displayed in the **Data info** window when the other body is not visible.

Troubleshooting update

Symptoms

QTM is old and you want to update the software.

Resolution

- Look for the latest software on this web page http://www.qualisys.com/login.html. Log in with your user name and license id, which are located on the inside of the front cover of the QTM installation CD. The user name and license id can also be found in QTM, go to **About Qualisys Track Manager** on the **Help** menu.

Symptoms

QTM needs to update the firmware and you cannot find it on your computer.

Resolution

- The firmware should be located in the **Oqus_firmware** or **MCU_firmware** folder in the **Qualisys Track Manager** folder. If it is not there install the latest QTM version again to get the correct firmware.

Troubleshooting Oqus wireless

Symptoms

Cannot connect to a Oqus system on a wireless network.

Resolutions

- Check that the computer is connected to the correct wireless network (SSID).
- Because of Windows the computer can sometimes switch wireless network. This can happen if there is another wireless network which is set to connect automatically in the program used to manage wireless networks on the computer. The other network will be connected when the camera reboots and its network disappears temporarily. The solution is to disconnect all other wireless networks before connecting to the Oqus wireless network.
- Check that the network interface has the correct settings, see chapter “Network card setup” on page 341.
- Check that the wireless Oqus camera has a small antenna in the display. If not connect the system to a wired network and run the QDS wizard, see chapter “Oqus wireless camera setup wizard” on page 335.
- The wireless connection can be lost when switching the active wireless camera in a system. This is because Windows will keep its connection to the previous ad hoc SSID even if the new camera has the same SSID. The solution is to go in manually and disconnect and delete the old SSID and then connect to the new.

Symptoms

When starting a new measurement the **Waiting for Oqus cameras to connect** dialog is never closed and it reports a master conflict.

Resolutions

- There are two separate wireless networks with the same SSID. This will result in a master conflict, because the synchronization is only done over the wired interface. Change the SSID of one of the systems.

Symptoms

The wireless network on the computer reports a communication speed less than 54 Mbit/s.

Resolutions

- Check that the wireless network on the computer is compatible with the 802.11g standard. The maximum speed for the 802.11b standard is 11 Mbit/s.
- The communication speed drops if the signal strength is too low, i.e. with increasing distance between the computer and camera.
- The communication speed can drop to 11 Mbit/s if there is more than two units (computer and camera) connected to the network. Disconnect one of the computers.

Troubleshooting other

Symptoms

There are less analog channels displayed in the **Data info** window than there should be.

Resolution

- Check that all of the analog channels have been activated on the **Analog board (...)** page in the **Project options** dialog.

Symptoms

The **Status** bar shows the message ”No analog trigger received” after a motion capture.

Resolution

- Check that the synchronization cable between the camera and the analog board is connected.

Symptoms

There is a **Data overrun** error for the USB-1616FS board even though the analog capture rate is below 2000 Hz.

Resolution

- Unplug the USB cable and power for the USB1616 board. Then restart the computer and connect the analog board again.

Symptoms

There is a **Too fast pacer rate** error for the USB-2533 board even though the analog capture rate is low.

Resolution

- Check the sync cable that is connected between the camera system and the analog board.

Symptoms

One or more trajectories are missing in the **Trajectory info** windows.

Resolution

- Retrack the file with new tracking parameters.
  Note: This will delete all processing that have been made to the capture file.

Symptoms

QTM crashes after several measurements.

Resolution

- Remember to close the QTM windows after each measurement, since every new measurement opens a new window.

Symptoms

The frame rate in preview mode is lower than 10 Hz.

Resolution

- Check that **Viewfinder** is not running in a 2D view window. The **Viewfinder** can lower the frame rate to less than 1 Hz depending on the settings.

- Close some **View windows**. The frame rate will be lowered by every **View window** that is open in preview.

Symptoms

You cannot open a new window.

Resolution

- There can be 30 **View** windows opened for a qtm-file, this also includes the **Plot** windows. If you have 30 windows opened, close one to open a new window.

Symptoms

The USB-2533 analog board is not detected in QTM or Instacal.

Resolution

- Check that the power cord is connected. When the power adapter is connected the board cannot get the power from the USB connector.

Appendix A: Oqus - Camera manual

Introduction

These chapters describes the Oqus camera from a technical perspective. It describes the mechanical properties as well as communication interfaces and accessories. In most configurations the camera is run in a multi-camera environment and controlled through QTM from Qualisys.

Configurations

The Oqus camera is designed to be adaptable to a large number of applications, ranging from the most demanding application requiring state-of-the-art performance to more cost sensitive OEM application. Its modular design provides possibilities for future upgrades and customer specific adaptations.

The configurations described in the chapters below are different accessories that are available for the Oqus camera. All other sections describe features common for all versions of the Oqus camera, except when explicitly stated otherwise.

Sensor

The Oqus camera is available with three different image sensors, with different key features. One provides exceptional resolution of 4.0 Mega pixels @ 180fps and another exceptional speed of 500 fps at 1.3 Mpixels. The last provides exceptional price/performance ratio with a VGA sensor running at 250fps.

Oqus 3- and 5-series cameras can be run up to 10000 fps by reducing the FOV and the Oqus 1-serie can be run up to 1000 fps. The table below shows how much the FOV must be reduced at different frame rates.

Frame rate [fps]	Oqus 1-series FOV	Oqus 3-series and 3+-series FOV	Oqus 3+-series FOV (high speed mode)	Oqus 4-series FOV	Oqus 5-series FOV
50	Full resolution (640x480, 307200 pixels)	Full resolution (1280x1024, 1310720 pixels)	Full resolution (648x512, 331776 pixels)	Full resolution (1696x1710, 2900160 pixels)	Full resolution (2352x1728, 4064256 pixels)
100	Full	Full	Full	Full	Full

200	Full	Full	Full	Full	Full (187 Hz)
250	Full (247 Hz)	Full	Full	Full	75%
500	50%	Full	Full	Full (485 Hz)	38%
1000	25%	50%	Full	49 %	19%
2000	N/A	25%	Full (1764 Hz)	24 %	9%
5000	N/A	10%	35%	10 %	4%
10000	N/A	5%	18%	5 %	2%

High-speed video

The Oqus camera can be equipped to capture full-frame, full-speed, full-resolution high-speed video. In this configuration the camera is equipped with a large buffer memory and a clear front glass to get the best possible performance out of the image capture. The table below shows the measurement length for different frame rates and resolutions when using a 1.1 GB memory and 8bits raw images. For other image or frequencies use the following formula "Maximum capture time" = 1207959552/ "X size"/ "Y size" / "Capture frequency".

Note: The 4-series camera is not sold with a high-speed video configuration.

Frame rate [fps]	**1280x1024**	**640x480**	**320x240**
50	18	76	307
100	9	38	153
250	3.6	15	61
500	1.8	7.7	31
1000	N/A	3.8	15
2000	N/A	1.9	7.7
4000	N/A	N/A	3.8
8000	N/A	N/A	1.9

Optics

A large number of lenses are available for the Oqus camera. The native camera mount is C-mount but adapters to accommodate Nikon F-mount, Pentax K-mount, Minolta MD-mount and others are available. The standard camera is equipped with a lens providing approximately 40 degrees of horizontal field-of-view (HFOV). Contact Qualisys for available lens options that fit a specific Oqus camera series.

Communication

As standard, the Oqus camera is equipped with 100Mbit Ethernet for fast TCP/IP communication. As an option, the camera can also be equipped with 54Mbit WLAN (802.11b/g) for extended flexibility. See chapter “TCP/IP” on page A - 7 for more information.

Memory

The buffer memory of the Oqus camera can be selected from 144 MB memory up to 1.1 GB of memory for high-speed video camera application. Refer to the table below for available configurations. The recording times for markers stated in the table below, are calculated for frame buffer mode, meaning no data is fetched during the measurement. The recording times for images are calculated for uncompressed images.

The size of a marker frame in bytes is calculated through the following formula: 32 + 16 * "Number of Markers".

Memory size [MB]	Recording time - markers		Recording time - images	
	30 markers @ 30 fps	30 markers @ 500 fps	VGA @ 30 fps	VGA @ 500 fps*
144	2.7 h	10 min	16 s	1.0 s
288	5.4 h	20 min	32 s	2.0 s
576	11 h	39 min	65 s	3.9 s
1152	22 h	79 min	130 s	7.9 s

*Not available for Oqus 1-series.

Strobe light

As standard the Oqus camera is equipped with NIR LEDs that emits a light which is not visible to the naked eye. This is often preferred since the subject is not disturbed with strong visible light. However, in some applications it might be necessary to use a different wavelength on the light. Red light for example, is suitable when the Oqus camera will co-exist with a system that uses NIR light to communicate, for example 3D shutter glasses that is synchronized though NIR light. Cyan light is preferred when measuring under water, since longer wavelength is absorbed more by the water.

See chapter “How to change strobe unit” on page A - 7 for information on how to change the strobe.

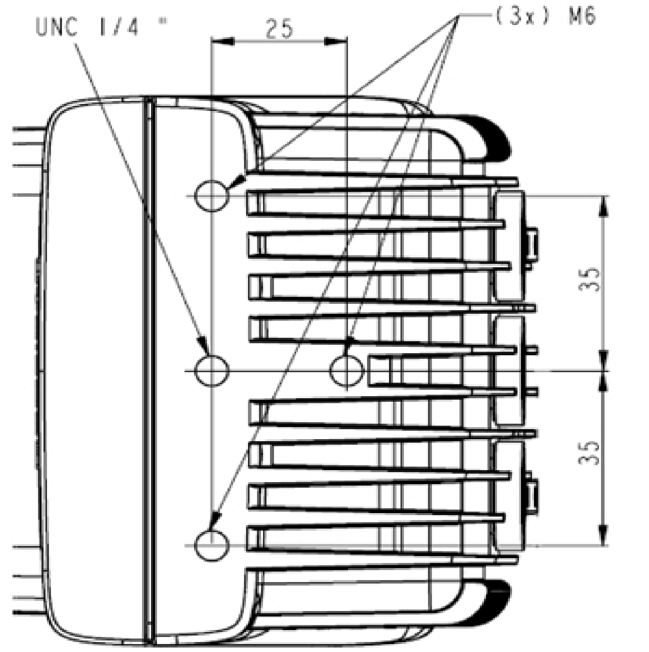
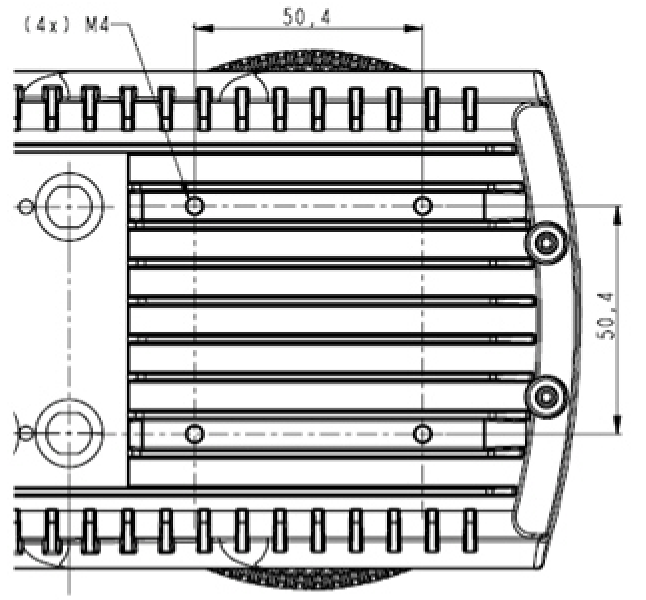
Environmental protection

Besides the standard housing, the Oqus camera can be delivered with housings that are protected from dust, water or magnetic interference. The standard version is suited for indoor usage while the water protected version is suited for outdoor usage. Two special versions of the housing are also available. One that is water-proof down to 10m and therefore suited for under-water measurements and one that is electro-magnetically shielded and therefore suited for use in an MRI-environment. Refer to the table below for the available versions.

Housing	Classification
Standard	Standard (IP30)
Water-protected	Water protected (IP67)
Underwater-proof	Waterproof - 10 m (IP68)
MRI	EMI shielded

Mounting

The Oqus camera has four standard mounting points on the bottom of the camera. One ¼" tripod mount and three M6 mounts for fixed mounting. For industrial mounts, "back-side mounting rails" that provides 4 x M4 mounting points on the back of the camera, is also available.

Mount	Type	Mounting	Drawing
Tripod	Standard	1xUNC ¼"	
Industrial	Standard	3xM6	
Back-side mounting rails	Accessory	4xM4	

Oqus technical specifications

Specifications of Oqus are listed in the chapters below .

Technical specifications

The two tables below show the specifications of the Oqus camera. The first table contains specifications that are the same for the all Oqus series. The second table show specifications that are specific for each Oqus series.

Camera output modes	Marker coordinates and high speed video*
Built-in camera display	128 × 64 graphical high contrast OLED
Camera body	Convection cooled, custom, die-cast aluminium
Camera size	185 × 110 × 125 mm (7.3 × 4.3 × 4.9 inches)
Weight including optics	1.9 kg (4.2 Ibs)
Camera protection level*	Water resistant IP67 housing available
Operating temperature	0-35 °C
Firmware	Upgradable from host computer
Position data noise level	+/- 1 camera units
Adjustable threshold	Yes
Frame buffer speed	12.9 Gbyte/second
Maximum frame buffer size	1152 Mbyte
Synchronization options	Internal 1ppm clock / external frequency output / external word clock input / SMPTE input
Cabeling	Bundled cable with Ethernet and power
Wired communication	Hubless daisy-chained Ethernet 802.3@100Mbps
Wireless communication*	WLAN 802.11b/g@54mbps
Power supply	Daisy-chained power from mains adaptor
Power	36-72 VDC, 10-16 VDC (battery) at 25 W maximum
Lens types*	SLR optics for Oqus 3+/4/5, c-mount for Oqus 1
Standard field of view	Options include 58, 47, 40 and 20 degrees
Strobe types supported*	Infrared

* Optional accessory/feature

	1-series	**3-series/ 3+-series**	**4-series**	**5-series**
CMOS sensor size (pixels)	640×480	1280×1024	1696x1710	2352×17
Marker position resolution (sub pixels)	41000 x 30000	82000 x 65000	110000 x 110000	150000 x 110000

Maximum frame rate at full resolution and field-of-view	247 fps	503 fps	485 fps	180 fps
Maximum frame rate at reduced field-of-view	1000 fps	10000 fps	10000 fps	10000 fp
Maximum video frame rate (full resolution)	247 fps	503 fps	N/A	180 fps
Maximum video buffer capacity (full resolution)	3800 frames	900 frames	N/A	290 fram
Maximum video buffer capacity (full resolution and frame rate)	15.2 s	1.8 s	N/A	1.6 s
Active filtering	Yes	Yes (only 3+)	Yes	No
High speed mode	No	Yes (only 3+, 648x512 at 1764 Hz)	No	No

Mechanics

The Oqus camera is made of a three-piece die-cast aluminum assembly. The finish is anodized and powder painted with "Structured black" and "Clear silver". Some versions of the Oqus camera comes in other colors: for example the MRI camera is white.

The Oqus camera weighs from 1.9kg (4.2lb) depending on the selection of optics. The physical size of the housing is 185 x 110 x 84 excluding the strobe.

Physical specifications

The following is the physical specifications of the Oqus cameras.

Weight	9kg - 2.4kg (4.2 lb - 5.3lb)
Physical dimensions	Without strobe: 185 x 110 x 84 With strobe: Depends on lens and Oqus-series.
Operating temperature range	0 - 35°C
Storage temperature range	-10 - 55°C

Optics and strobe

The native camera mount is C-mount but adapters to accommodate Nikon F-mount, Pentax K-mount, Minolta MD-mount and others are available. CS-mount optics cannot be used due to its shorter back focal lengths compared to the C-mount. Contact Qualisys for available lens options that fit a specific Oqus camera series.

How to adjust aperture and focus

All optics available to the Oqus camera have adjustable aperture and focus. In the standard camera the optics is easily accessible by turning the strobe part counter-clockwise. After adjusting aperture/focus the opening is closed by turning the strobe clockwise. In the IP-classified version the locking screws on the side must be loosened and the strobe unit pulled out.

How to change strobe unit

Follow this procedure to change the strobe unit.

1. Make sure the power is turned off and the strobe is in its closed position.
2. Remove the strobe unit.
    a. On a standard housing Oqus, turn the strobe counter-clockwise, placing it in the completely open position. Now, find the exit track and turn the strobe so that clicks in to the exit track. Pull the strobe carefully out of the housing.
    b. On an IP-classified housing, loosen the three locking screws on the side and pull out the strobe carefully. The gasket might make it difficult to pull the strobe out. Turning it side-to-side while pulling helps .
3. Disconnect the strobe cable. There is either a connector on the cable or three strobe connectors on the strobe unit. Always pull the connectors, not the cable!
4. Attach the strobe cable to the new strobe. For the connectors on the strobe unit make sure the red cable is attached to the pin marked '+' and that the black cable is attached to the pin marked '-'.
5. Re-attach the strobe unit
    a. On a standard housing Oqus, insert the strobe with the connectors facing down and the opening aligning with the opening on the camera. Align the tracks in the strobe unit with the thrust screws on the camera housing and simply push the strobe in.
    b. On an IP-classified housing, insert the strobe with the connectors to the left and align the holes on the side of the flash with the locking screws on the side of the camera housing. Then tighten the locking screws.

How to change lens

All lenses on the Oqus cameras can be change by the user by following the steps below.

1. Loosen the strobe unit by following step 1 and 2 in chapter “How to change strobe unit” above.
2. Simply unscrew the lens and replace with the new lens.
3. Fasten the strobe unit by following step 5 in chapter “How to change strobe unit” above.

TCP/IP

The Oqus camera can communicate with a host computer through three different interfaces.

- TCP/IP over 100Mbps Ethernet
- TCP/IP over 54Mbps WLAN

Ethernet (802.3) and Wireless LAN (802.11) are both physical transmission media over which TCP/IP can be carried.

All Oqus cameras are equipped with daisy-chained Ethernet which means that a virtually unlimited number of cameras can be connected in a chain without the requirement of an external hub or switch. It is also possible to connect the cameras in a star network with external gigabit switches if that for some reason is beneficial for the camera setup.

Ethernet standard specifies that the maximum cable length between each node to 100 m. However we recommend keeping it below 60 m to optimize the performance.

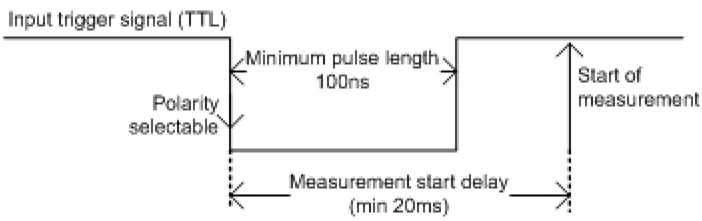
One Oqus camera in a system can be equipped with WLAN for wireless communication. Then the camera with WLAN communicates wireless with the host computer. The rest of the cameras are connected through Ethernet in a daisy-chain as usual.

Digital and analog IO

The Oqus camera provides several inputs and outputs, both digital and analog.

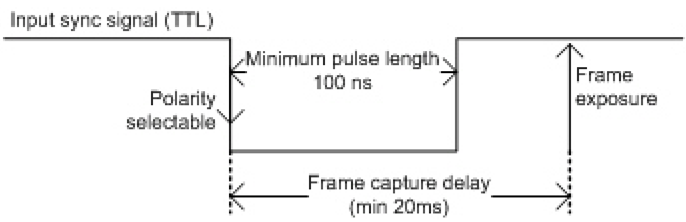
Trigger input

The camera can receive a trigger signal from an external TTL source. Depending on the selected mode, the external trigger can either start a pending measurement or switch from pre- to post trigger measurement. In a multi-camera environment, the trigger can be connected to any camera in the system. The time between the signal is asserted and the action in the camera can be selected on the **Timing** page in the **Project options** dialog. The polarity is selectable on the same page.



Synchronization input

The Oqus camera can be synchronized by an external source and configured to except almost any type of sync signal. The polarity is selectable as well as the delay between the sync signal and the exposure. If the external sync signal is periodical, the camera can be configured to lock-on to the periodic signal and the exposure can be freely moved between 0 and 360 degrees. All settings are done on the **Timing** page in the **Project options** dialog.

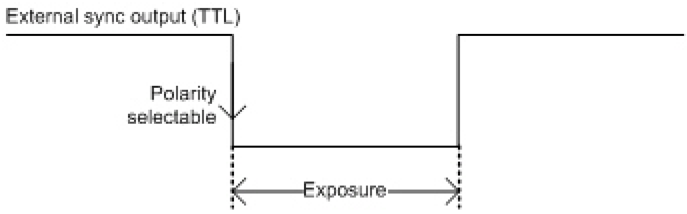


Synchronization output

The Oqus camera has an output TTL signal that is hard-wired to the exposure. In the default mode a pulse is sent for each frame that is captured by the camera and the result is a pulse train during the capture. The polarity can be selected and the signal is asserted when the exposure starts and returns to default when the exposure ends.

The sync out signal can also be modified to use other frequencies and pulse durations. All settings are done on the **Timing** page in the **Project options** dialog.

This signal is the recommended option to synchronize external equipment to the camera exposure. The external equipment must be able to trigger on an edge to achieve the best synchronization.



Control connections

The connections to the control port are all done with short splitter cables. They all have BNC connectors where BNC cables can be connected to extend the length. There are three different splitter cables.

Sync out/Sync in/Trig splitter
This splitter has three connectors. One BNC female for Sync out, one BNC female for Sync in and one BNC male for Trigger in.

Sync out/Trig splitter
This splitter has two connectors. One BNC female for Sync out and one BNC male for Trigger in.

Sync in splitter
This splitter has one connector, a BNC female for Sync in.

Definition of Oqus types

The Oqus type is the number on the badge above the display. This number (ABCd) identifies the main feature of a camera according to the following list:

Letter	Features	Code	Option
A	Oqus type - Sensor	1	0.3 Mpixel
		3	1.3 Mpixel
		4	3 Mpixel
		5	4 Mpixel
B	Oqus type - High-speed	0	None
		1	Available
C	Oqus type - WLAN	0	None
		1	WLAN
d	Oqus type - Protection	N/A	Standard
		p	IP67
		m	MRI
		u	Underwater

An Oqus camera can for example be called 310p, which is a Oqus with 1.3 Mpixel sensor, High-speed and IP67 protection.

Note: It will say PLUS under the number if the camera is a 3+-series camera.

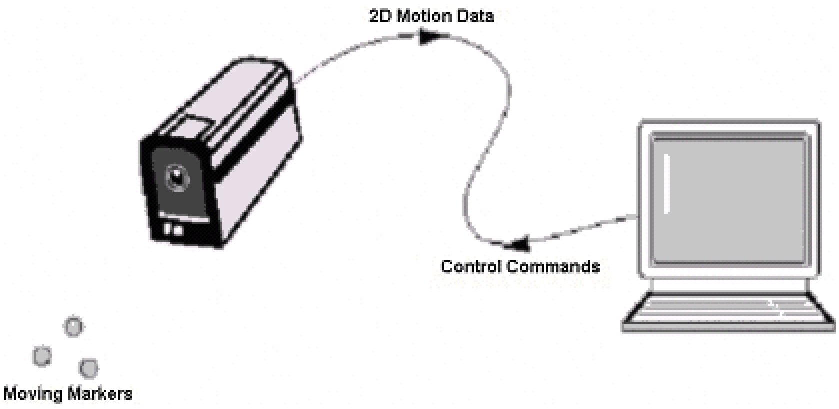
Appendix B: ProReflex MCU - Camera manual

Introduction

With the ProReflex MCU (Motion Capture Unit) several steps in the data processing of motion capture have been combined in one single unit: flash exposure, image retrieval, image processing and data transfer.

The high precision and high sampling rate of the unit, combined with the possibilities to control the measurement parameters, offers great prospects to optimize the unit for any particular measurement situation.

The advanced hardware design matched with the continuous upgrade possibilities will assure that the ProReflex MCU remains state of the art long after the purchase.



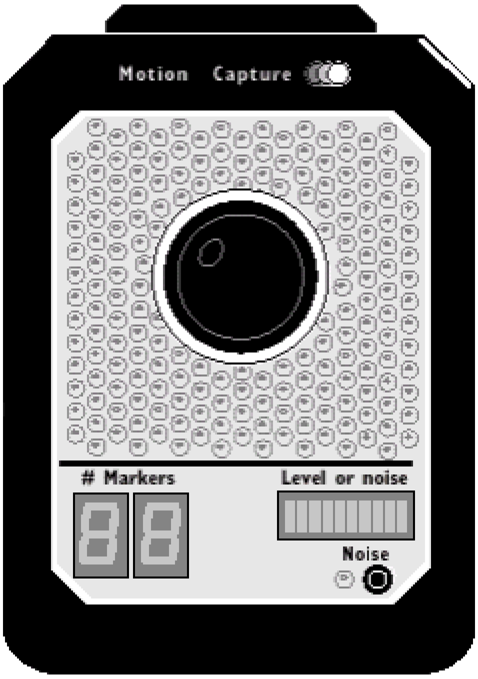
Hardware overview

The basic principle of the ProReflex MCU is to expose reflective markers to IR light and to detect the light reflected by the markers. The 2D image of the markers is processed by the MCU and the 2D coordinates of each marker are sent as a data stream output.

A system of several units can be configured to perform multi-camera measurements. 2D data from each camera in such a system can be retrieved simultaneously. If 2D data from more than one camera is combined, it can be used to calculating the 3D position of markers.

The setup and operation of the MCU camera system are controlled from a measurement computer and so is the control of the internal software of the MCU.

ProReflex MCU front view



The front view of the ProReflex MCU is shown above.

The **# Markers** display shows the number of markers detected and accepted by the MCU. When the MCU is used in the default mode, the display is continuously updated during the measurement. The range of the display is limited to 99 markers, even though the MCU can detect more than 99 markers.

Note: In certain measurement modes (grayscale mode, segment mode and when measuring with Long range active markers) the number of markers cannot be displayed, so the front shows '[]' to indicate that it is measuring.

The display is also used as a display for error codes, which are displayed as two numbers separated by a dot (e.g. '0.1'). It can also show camera ID or buffer fill level upon request from host.

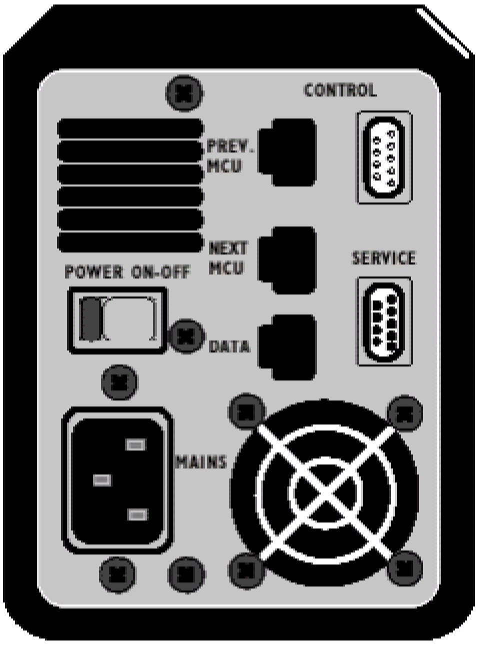
At power up, the **# Markers** display executes a start up sequence. First it briefly illuminates '88', to show that power is supplied to the unit. Then the display presents the ID number of the MCU. Finally it shows two horizontal bars, to indicate that the camera is ready to communicate with the measurement computer and other MCUs.

The **Level or noise** indicator can be used in either **Level** or **Noise** mode. In the **Level** mode it indicates the level (intensity) of the most intense marker detected by the MCU. The button labeled **Noise** is used for toggling the bar-graph indicator between **Level** and **Noise** mode.

Note: *The **Noise** mode is not yet implemented.*

Around the lens there are IR light emitting diodes (LED), which can send IR flashes. The LEDs are grouped into 3 rectangular shaped groups; inner, middle and outer, where the inner group LEDs is located closest to the lens.

ProReflex MCU rear view



The rear view of the MCU is shown above. The following connections are available on the MCU:

Data port
Connection for the standard RS422 communication link between the measurement computer and the master MCU.

Service port
Connection for maintenance of the MCU (e.g. download of new MCU software).

Next MCU / **Prev. MCU** ports
Connections to connect several MCUs in a chain.

Control port
Connection for control of the MCU measurement from an external source. In multi-camera systems it is only the **Control** port of the master MCU that is used.

POWER ON-OFF switch
Switch on and off the MCU. In multi-camera systems, the power switch of the master MCU is used to power up all MCUs.

MAINS connector
Connection for power supply to the unit. Each unit in a system is supplied individually and the voltage of the supply must be between 100 and 250 V AC.

The fan at the rear of the MCU supplies the MCU with cooling air. Make sure that the inlet and outlet of cooling air are not blocked while the unit is in operation. Blocking the flow of cooling air may cause hardware damage. The access door on the top of the MCU should be closed at all times to allow the fan to operate correctly. The only exception is when the aperture or focus setting is adjusted.

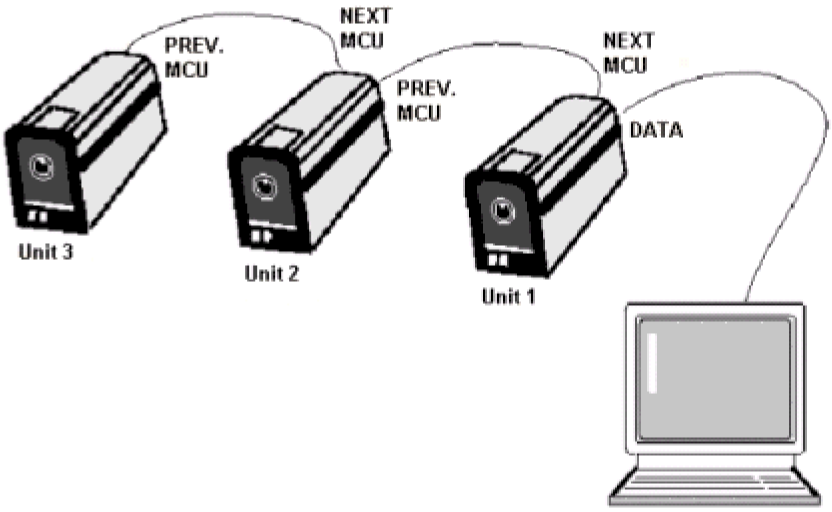
Setting up the system

Up to 16 MCUs can be used in a 3D motion capture system, where the units are connected to each other with the **Next MCU** and **Prev. MCU** ports. The entire camera system is then connected to the measurement computer with the **Data** port of the master MCU. Use the cables that are distributed with the system to connect the system.

To set up the system, go through the following steps:

1. Connect the data cables (blue) between the MCUs. The IDs of the cameras can be set in any order.
2. Connect the RS 422 cable (yellow) between the **Data** port of the master MCU and the RS 422 port on the serial communication board in the measurement computer.

   Note: If a regular COM port is used connect the RS 232 cable (black RJ45 connector) between the **Data** port of the master MCU and the COM port.
3. Set all power switches to off and connect all units to power supplies.
4. Switch on the power supply of the master MCU. All other units will then be automatically switched on.



Arranging the cameras

Once the system has been properly set up, the cameras must be arranged to the current measurement setup. When arranging the cameras, it is best if they are in operating mode. Start the measurement computer and the QTM software. Connect camera system to QTM on the **Connection** page in the **Project options** dialog, see chapter “Connection of the camera system” on page 227. Open a new file with a **2D view** window.

Arrange the cameras to cover the entire measurement volume of the wanted motion. It is advisable to mark the limits of the measurement volume with markers and use the **2D view** window of the software to make sure that all markers are seen by all cameras. For guidelines on how to arrange the cameras see chapter “Camera positioning” on page 250.

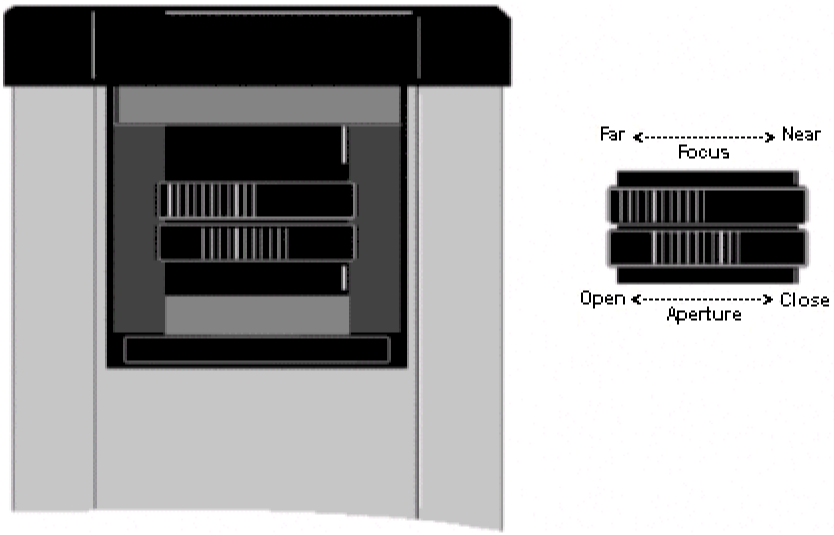
Note: The number of markers seen by each camera is also shown on the display at the front of each camera.

Frequency

Before adjusting the aperture and the focus of the cameras, the frequency of the measurement must be set. This is done in the QTM software on the **Camera system** page in the **Project options** dialog. The reason for setting the frequency first is that the camera resolution and light sensitivity are frequency dependent.

Aperture and focus

The lens of the MCU can be reached through the access door at the top of the unit. Two parameters of the lens can be adjusted: aperture and focus. It is recommended to first set the focus and then the aperture according to the following sections. The focus is set on the adjustment ring closest to the front, while the aperture is set on the one closest to the rear.

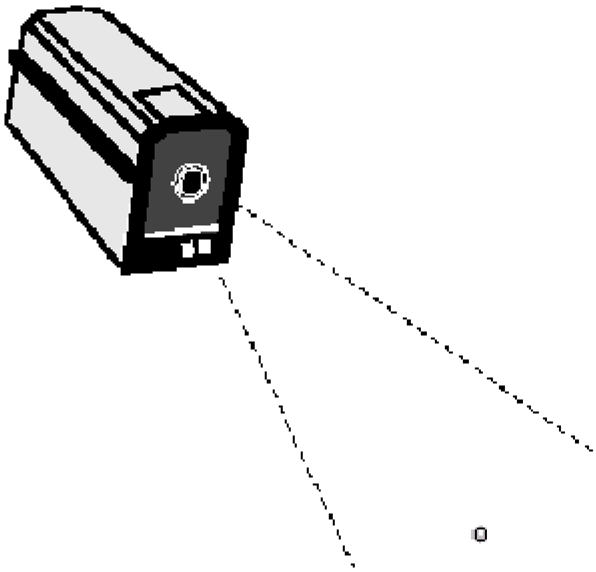


Setting the focus

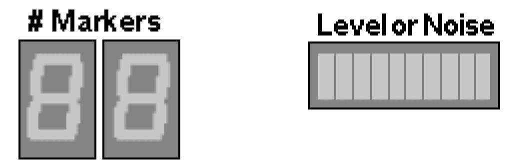
When the camera system has been arranged, the focus of each camera is set by going through the following steps:

1. Make sure that the correct frequency is set for the camera system and place a small marker at the furthest possible distance from the camera, within the measurement volume. In the table below, suitable marker sizes for setting the focus are shown for different distance ranges.

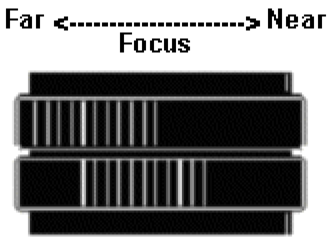
Marker size	Distance range
4 mm	< 1m
7 mm	1 – 2 m
12 mm	2 – 4 m
19 mm	4 – 6 m
30 mm	6 – 8 m
40 mm	> 8 m

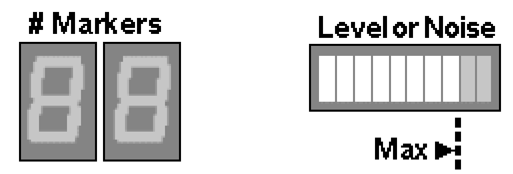


2. Adjust the aperture until the **# Markers** display shows '1' marker and approximately seven to eight bars of the **Level** indicator are filled.



3. Adjust the focus ring until the optimum focus is achieved. It is when the **Level** indicator reaches its maximum value for this setup. The maximum level is therefore not equal to 10 bars on the **Level** indicator.

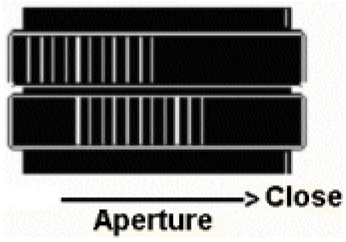




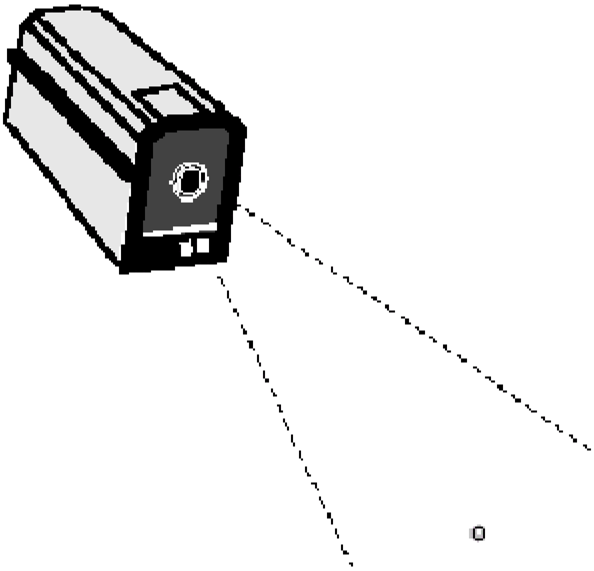
Setting the aperture

After the focus has been set, the aperture of the camera is set by the following procedure:

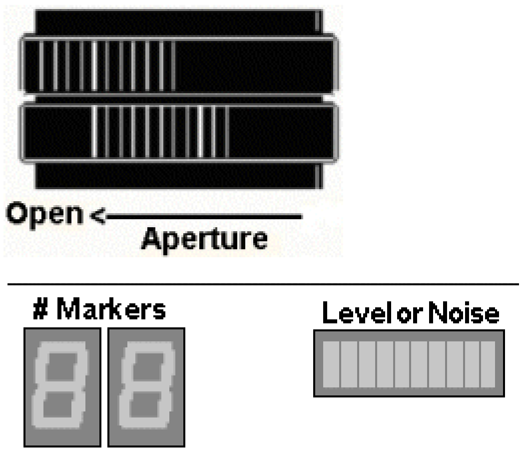
1. Close the aperture completely by turning the aperture ring. No bars on the **Level** indicator will be lit.



2. Place a marker, with the same size as those that will be used in the measurement, at the furthest possible distance from the camera, within the measurement volume.



3. Open the aperture until the **# Marker** display shows '1' marker and approximately eight bars of the **Level** indicator are filled.



MCU functional specification

The functions of the ProReflex MCU are listed below, more detailed information is found in the chapters “Hardware signals specification” on page B - 10 and “MCU technical specifications” on page B - 13.

Frame rate (Internal timebase)

The MCU can operate in the frequency range of 1 to 1000 Hz, depending on the model. In a ProReflex MCU camera system all MCUs operate at the same frame rate, regulated by the master MCU. An internal clock in the master MCU keeps the frame rate. This feature is controlled via a host command.

Note: A frequency dependent automatic line summation is applied in each MCU to optimize the image processing at high frequencies.

Synchronization of external equipment

An external unit can be synchronized with ProReflex MCU using the **External sync output** pin of the **Control** port on the master MCU. The signal on the **External sync output** pin is a TTL signal coinciding with each exposure initialization.

Note: The actual image exposure of the camera is delayed 400 µs compared with the exposure initialization.

External trigger

An external trigger can be connected to the camera system to control the start of the motion capture. The external trigger is connected to the **External trig input** pin of the **Control** port on the master MCU. This feature is controlled via a host command.

Pretrigger

A pretrigger can be connected to the MCU to control the collection of pretrigger frames. The pretrigger is connected to the **External sync input** pin of the **Control** port on the master MCU. This feature is controlled via a host command.

External timebase

An external timebase device can be connected to the MCU system to control the frame rate of the camera system. The external timebase device is connected to the **External sync input** pin of the **Control** port on the master MCU. This feature is controlled via a host command.

MCU operation modes

The MCU can be run in three different modes: **Immediate** mode, **Frame buffer** mode and **Segment buffer** mode. The modes can be used to optimize the measurement system to the current measurement conditions. The modes are controlled via host commands.

Immediate mode

In **Immediate** mode the data collected by the MCU is continuously retrieved by the measurement computer during the measurement. If the data in the output buffer of the MCU is read by the measurement computer at the same rate as it is retrieved by the MCU, there is no upper limit for the total amount of data collected in a measurement.

When in **Immediate** mode the MCU will process both the data retrieval and the data transfer to the measurement computer simultaneously. If the markers at the bottom of the FOV are not calculated, the **Frame buffer** mode or **Segment buffer** mode can be used instead.

Frame buffer mode

In **Frame buffer** mode there is no communication between the measurement computer and the MCU during the data retrieval. Consequently all of the image data is stored as calculated frames in the MCU frame buffer until the end of the measurement. In this way the MCU can run on a higher frame rate compared with the **Immediate** mode.

The total amount of data is however limited by the buffer size of the MCU frame buffer. The maximum measurement time, which is possible to use, is therefore a function of the frame rate and the number of markers. These parameters can be weighted against each other in order to optimize the data processing of the MCU according to the current measurement situation.

Segment buffer mode

In **Segment buffer** mode the processing capabilities of the MCU is even more focused on data retrieval optimization. In this mode the data read from the CCD of the MCU is stored in the segment buffer before marker calculations are performed. The calculation of marker coordinates is not performed until the data is requested to be retrieved by the measurement computer.

The **Segment buffer** mode is useful in high-speed measurements to optimize the data retrieval. However, because the segment data is larger than the data after the calculation of marker coordinates this option is preferably used for measurements with a short duration.

Max number of markers limit

The maximum number of markers allowed in each frame can be set. Since the MCU scans the FOV from top to bottom the lowest placed markers will be rejected if the number of markers in the image exceeds the max number of markers specified. This feature is controlled via a host command.

Marker size limits

The minimum and maximum size of the markers can be set in order to reject the transfer of false markers from the cameras. The marker size is given in per mill of the diagonal FOV. This feature is controlled via a host command.

Circularity

A circularity check can be used in the MCU to delete markers that are not round enough. The circularity is given by the ratio of the size of the marker in X and Y directions. It is set in percent, where 0 means any shape and 100 means perfect circularity. This feature is controlled via a host command.

Use of active markers

The MCU can be used with active (IR light emitting) markers, which are either synchronized or continuous.

If synchronized active markers are used the flash of the markers is triggered by the MCU flash. In this case the MCU flash is modulated at 38 kHz. Passive markers and synchronized active markers cannot be mixed in the same measurement. The settings to use the MCUs with synchronized active markers are controlled via a host command.

Continuous active markers transmit IR light constantly. They can therefore be used together with passive markers. At such a capture the ProReflex MCU should be set to use passive markers.

IR flash control

The flash LEDs at the front of the MCU are divided into three groups with respect to their positions relative the camera lens and are therefore called: inner group, middle group and outer group. The groups can be used to optimize the flashes for different distances. This feature is controlled via a host command.

The following selections of LED groups are recommended, with respect to the distance between camera and markers:

Distance	**LED group**
0 – 2.5 m	Inner
0 – 4 m	Inner + Middle
0 – 30 m	Inner + Middle + Outer

Baud rate

The baud rate to the MCU is set to the maximum rate that is possible for the system. The maximum rate depends on the measurement computer.

Default MCU startup settings

At the start up of the camera system every MCU will automatically find its position in the chain. Furthermore, every MCU initializes itself according to:

Frame rate (Internal timebase)	50 Hz
Line summation	1 (No summation)
External trigger	Off
Pretrigger	Off
External timebase	Off
Operation mode	Immediate
Gate enable	Off
Max markers/frame	20
Low limit marker size	1
Max limit marker size	100
Circularity	0
Active markers	No
LED groups used	All
Host communication baud rate (kbit/s)	19.2

Hardware signals specification

The following chapters describe the signals on the ports of the MCU, i.e.**Next MCU**/**Prev. MCU**, **Data**, **Service** and **Control** port.

Next MCU / Prev. MCU ports

The **Next MCU** and **Prev. MCU** connectors provide a high speed, full duplex communication link between the MCUs in a camera system.

Data port

The pins of the RS-422 **Data** port are defined according to the following table:

Pin	Abbr.	Signal	In/Out
pin 1	HSKi	Handshake	In
pin 2	CLK	Clock	Out
pin 3	RXD-	Receive Data (-)	In
pin 4	GND	Ground	
pin 5	TXD-	Transmit Data (-)	Out
pin 6	RXD+	Receive Data (+)	In
pin 7	GPI	Reserved	
pin 8	TXD+	Transmit Data (+)	Out

Handshake (data port)

The handshake of the data port is implemented with the **HSKi** signal. The **HSKi** signal must be forced high by the measurement computer to make the MCU send data. The transmission will stop after 1 or 2 bytes when the **HSKi** signal is forced low.

Service port

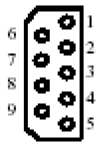
The pins of the RS-232 **Service** port are defined according to the following table:

Pin	Abbr.	Signal	In/Out
pin 1	NC	Reserved	
pin 2	RD	Receive Data	Out
pin 3	TD	Transmit Data	In
pin 4	DTR	Data Terminal Ready	In
pin 5	GND	Ground	
pin 6	DSR	Data Set Ready	Out
pin 7	RTS	Request To Send	In
pin 8	CTS	Clear To Send	Out
pin 9	NC	Reserved	

Handshake (service port)

The handshake of the service port is implemented with the **CTS** signal. The **CTS** signal must be forced high by the measurement computer to make the MCU send data. The transmission will stop after 1 or 2 bytes when the **CTS** signal is forced low.

Control port



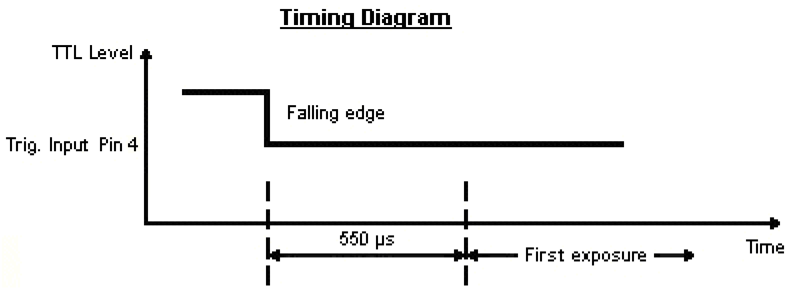
The pins of the RS-232 **Control** port, see image above, are defined according to the following table:

Pin	Function	In/Out
pin 1	Reserved	
pin 2	Reserved	
pin 3	Reserved	
pin 4	External trig input	In
pin 5	Ground	Out
pin 6	+5V Supply	Out
pin 7	External sync input	In
pin 8	External sync output	Out
pin 9	Ground	Out

The signals in and out from the **Control** port are TTL level compatible.

Below follows explanations of the pin signals.

Pin 4, External trig input
Trigger input signal. It is a TTL signal and active low, which means that the cameras start on the falling edge. The first image exposure occurs 550 µs after the falling edge (see figure below).



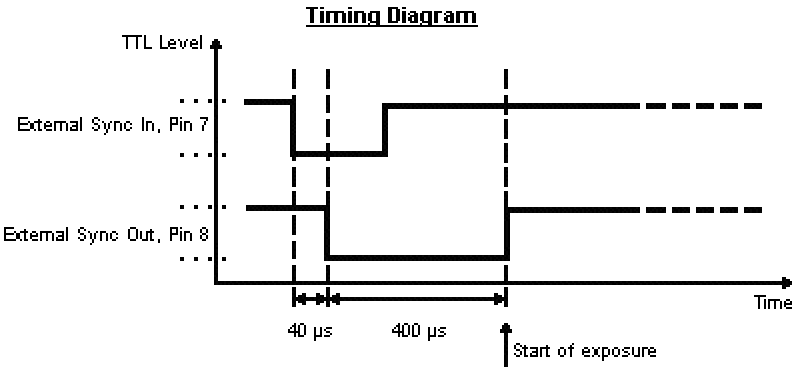
Pin 5 and Pin 9, Ground
All signals of the **Control** port are referenced to this ground.

Pin 6, +5V supply voltage
Max current output: 100 mA.

Pin 7, External sync input
Input signal of the external sync. It is a TTL signal and active low. The image exposure of the cameras is started 440 µs after the falling edge of this input signal (see figure below).

Pin 8, External sync output
Output signal of the external sync. It is a TTL signal and active low. The falling

edge coincides with each exposure initialization. The actual image exposure of the camera is delayed 400 µs with respect to the exposure initialization (see figure below).



Note: The time lag on the **External sync output** pin is the same for all frequencies. Exposure duration is, however, frequency dependent.

MCU technical specifications

Specifications of the ProReflex MCU are listed in the chapters below .

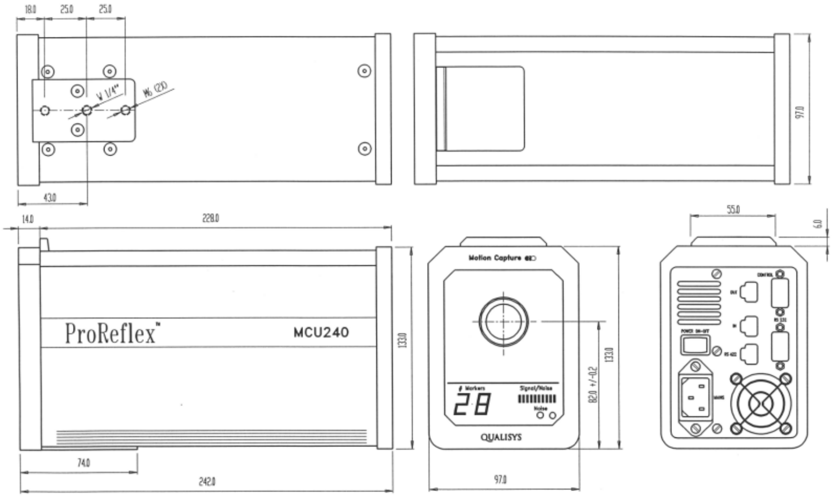
MCU specifications

The following specifications are independent of the mode of the MCU:

Camera model	**Sampling rate**
ProReflex MCU120	1-120 Hz. Resolution 1 Hz
ProReflex MCU240	1-240 Hz. Resolution 1 Hz
ProReflex MCU500	1-500 Hz. Resolution 1 Hz
ProReflex MCU1000	1-1000 Hz. Resolution 1 Hz
Marker types	**Passive**: Reflective tape **Active**: IR-emitting light sources
Marker type	**Measurement distance**
Passive markers	< 2.5m Inner LED group. < 4.0 m Inner + middle LED groups < 30 m All LED groups
Active markers	Dependent on marker intensity and ambient conditions
Horizontal FOV	Lens dependent, typically 43° with f6 lens. (For MCU500 and MCU1000 the FOV is limited by 75 % and 50 % respectively)
Operating temp. range	5 – 35 °C
Relative humidity	0 - 100 % (Non condensing)
Cable lengths	10 m standard between MCUs
Frame buffer size	CPM3: 805 Kbytes CPM31: 1806 Kbytes
Segment buffer size	CPM3: 808 Kbytes CPM31: 1808 Kbytes

MCU-MCU synchronization	±5 µs
MCUs in one system	Max 16 MCUs

Mechanics



Weight
2.4 kg

Mounting support
The lens is fixed to an L-shaped aluminum profile. At the base of this profile there is a UNC 1/4" for mounting on standard tripods. There are two M6 holes for bolts for precision mounting.

Lens mount
Matches C mount lenses.

Normal lens
f6

Standard lenses
f6, f12, f16, f25 and f50.
All of the lenses have aperture and focus controls.
For further information see “ProReflex lens types” on page B - 17.

Image area
4.87 x 3.67 mm

Picture elements
658 (Horizontal) x 496 (Vertical)

Infra Red Strobe
250 IR diodes divided into 3 Groups.

Inner group: 40 diodes.

Middle group: 80 diodes.

Outer group: 130 diodes.

The spectral sensitivity curve for sensor, lens and filter is a narrow pass band around the wavelength for the strobe.

Galvanic isolation
All MCU-MCU connections are galvanically isolated.

MCU service and maintenance

The following chapters describe how to check the firmware of the MCU.

MCU firmware

The internal firmware of the MCU can be installed or upgraded from the measurement computer with the QFI software, see “Qualisys Firmware Installer (QFI)” on page D - 1 for detailed information. To download a certain firmware open that folder in the **MCU_ firmware** folder in the **Qualisys Track Manager** folder and use the QFI.exe that is located in that folder. The firmware transfer is either done through the **Service** port at the rear panel of the MCU, or through the standard data port.

To upgrade the firmware in a single camera through the service port, connect the **Service** port of the MCU to a serial communication port of the measurement computer, **NOT** the Brainboxes board. A pin to pin cable can also be used to upgrade the software. Once connected, start the upgrade program (QFI), select service port installation and follow the instructions. This option has to be used for MCUs with a firmware version which is earlier than 7.0.

To upgrade the firmware in all cameras connected to a system, start the upgrade program (QFI), select data port installation and follow the instructions. This option only works with cameras that already have a firmware version which 7.0 or higher.

Note: Make sure that all MCUs in the ProReflex MCU system use the same firmware version and that the measurement computer software is compatible with the current MCU firmware version.

Unit ID

Each camera must have a unit ID, from 1 - 32. If all cameras in the system have a unique serial number set, the IDs are automatically set when the host software locates the system. The ID can also be set manually via the service port with QFI, see “Qualisys Firmware Installer (QFI)” on page D - 1.

Error codes

The table below shows the error codes shown in the **# Markers** display of the MCU.

Code	Description
0.1	Initialization Failed
0.2	Unknown command
0.3	Invalid data for **c** command (circularity)
0.4	Invalid data for **f** command (frame rate)
0.5	Invalid data for **G** command (go)
0.6	Invalid data for **l** command (line sum)
0.7	Invalid data for **L** command (lower limit)
0.8	Invalid data for **M** command (max markers)
0.9	Invalid data for **T** command (transmit)
0.A	Invalid data for **U** command (upper limit)
0.B	Invalid data for **m** command (mode)
0.C	MCU is not in grayscale mode

0.D	MCU is not in normal mode
1.1	Invalid character
1.2	Invalid command length
1.3	Invalid command address
1.4	Invalid command code
1.5	Invalid command data
1.6	Invalid camera ID
1.7	Cannot write serial number, measurement running
2.1	Not able to initialize drivers
2.2	Transmission failed
2.3	Unable to register abort flag
2.4	Invalid baud rate
3.1	Frame buffer overflow
3.2	Frame buffer underflow
3.3	Frame size too large
3.4	Illegal size or start address of FB memory
3.5	Frame buffer allocation
3.6	Invalid frame buffer state
4.1	Illegal YBase value
4.2	Illegal number of YBase values
4.3	Illegal Y value
4.4	Illegal HW FIFO pixel number value
4.5	Illegal HW FIFO intensity value
4.6	Illegal HW FIFO A/D value
4.7	Expected end of frame
4.8	FIFO read time out
4.9	Overflow
4.A	Initialization of segment buffer failed
5.1	Illegal YBase value
5.2	Illegal pixel number
6.1	Unable to store new frame info. SW FIFO overflow
6.2	No frame info to deliver. SW FIFO underflow
6.3	Invalid state at 'Acquisition Prepare'
6.4	Unknown sync. in CA register
7.1	Illegal number of lines in line summing
7.2	Initialization of flash timer failed
7.3	Last image is still exposing when new starting.
7.4	Image exposing when setting exposure time
7.5	Illegal line sum with active markers
7.6	Error in PIC communication
7.7	PIC version does not support active markers
7.8	PIC update not allowed
7.9	ICI invalid mode
8.1	Not able to initialize temperature measuring
8.2	CCD chip temperature too high
8.3	Error measuring temperature
9.1	Communication with PIC-Processor failed

A.1	Firmware upgrade initialization failed
A.2	Invalid mode
A.3	Slot number out of range
A.4	Error loading application
A.5	Error programming application
A.6	Error verifying application
A.7	Error loading tfq file
A.8	Error programming EPLD
A.9	Error verifying EPLD
A.A	Error loading EPLD
A.B	Invalid exposure mode
B.1	Not in grayscale mode
B.2	Invalid image size
B.3	Buffer not allocated
B.4	Cannot allocate buffer
B.5	No data in HW FIFO
B.6	Wrong VA response
B.7	Last image is still exposing when new starting.

Note: Error codes 1.1, 1.2, 2.1, 2.2, 2.4, 4.1 - 4.9, 5.1, 5.2, 8.1 - 8.3 and 9.1 are not implemented or displayed.

ProReflex lens types

The table below shows data of lenses that can be used with the ProReflex MCU.

Manufacture	Focal length (mm)	Horizontal FOV (°)	Vertical FOV (°)	Horizontal FOV/Distance (%)	Vertical FOV/Distance (%)	MOD (m)	Dimensions (mm)
Goyo 80GM26014MC	6	42.9	33.0	78.5	59.2	0.2	35 x 34.0
Goyo 80GM38013MC	8	28.8	22.4	51.3	39.6	0.2	29 x 34.0
Goyo 80GM21214MC	12	23.0	17.2	40.6	30.2	0.3	29 x 29.5
Goyo 80GM31614MC	16	15.4	11.6	27.0	20.3	0.4	30 x 24.5
Goyo 80GM32514MC	25	9.7	7.5	16.9	13.1	0.5	30 x 24.5
Goyo 80GM35018MC	50	5.1	3.7	8.9	6.4	1.0	35 x 37.0

MC = Metal

Image format = 1/3 inch

MOD = Minimum Object Distance

Appendix C: Serial communication devices - Technical specification

Introduction

The Qualisys Track Manager software uses serial communication devices of the measurement computer to communicate with the camera system. QTM can use the following devices for the communication:

- Built-in COM ports
  Max baud rate 115.2 kbit/s

Windows XP

- Brainboxes PCI Dual Port RS422/485 [CC-530] board
  Max baud rate 2000 kbit/s
- Brainboxes PCMCIA Single Port RS422/485 [PM-120] board
  Max baud rate 921 kbit/s
- Sealevel PCI ULTRA 530.PCI [7101] board
  Max baud rate 2048 kbit/s
- Sealevel Low Profile PCI ULTRA 530.LPCI [7106-SN] board
  Max baud rate 2048 kbit/s
- Sealevel PCMCIA ULTRA-PC-SIO-485 [3602-SN] board
  Max baud rate 921 kbit/s

Windows 7

- Brainboxes PCIe 1xRS422/485 1MBaud Opto Isolated [PX-387] board
  Max baud rate 921 kbit/s
- Brainboxes ExpressCard 1xRS422/485 1MBaud [VX-023] board
  Max baud rate 921 kbit/s

To avoid long download times from the camera system it is recommended that one of the communication boards is used, at least when there are more than two cameras in the system.

Use the drivers supplied with the QTM installer for the Brainboxes and Sealevel boards. Other drivers have not been tested with QTM and may therefore not work. Sealevel have drivers for Windows 7, but these have not been extensively tested. And because Sealevel have not got ExpressCard for the laptop, Qualisys will only support Brainboxes on Windows 7.

Installing the serial communication device

Depending on what you want to install follow the instructions in one of the chapters below.:

Built-in COM ports

For the built-in COM port no installation is needed. If there are problems with this port, see the manual for the measurement computer.

PCI and PCI express boards

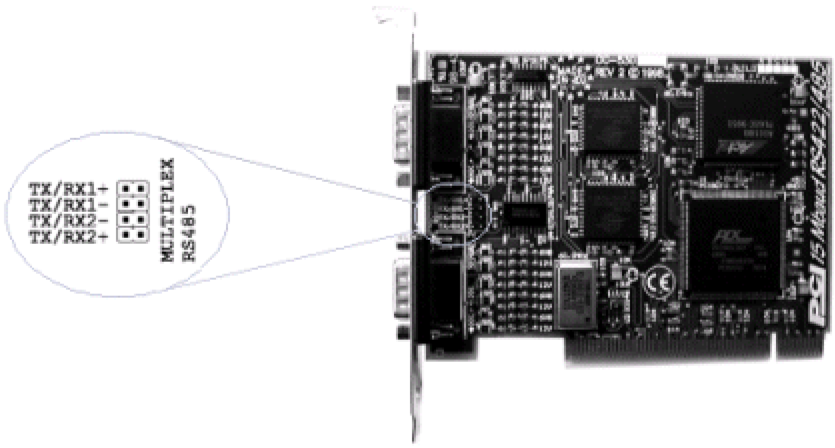
The PCI and PCI express boards can be installed in desktop computers. In the following chapters there is a description of how to install the boards.

PCI board configuration

The configuration differs between Brainboxes and Sealevel boards see chapters below.

Brainboxes PCI Dual Port RS422/485 [CC-530]

The Brainboxes PCI board requires no hardware configuration for the communication with QTM. I.e. the jumpers, for RS485 multiplex mode, on the PCI Dual Port RS422/485 board should be open, which is the factory default.



The IRQ and DMA setup is done by Windows plug and play at the startup of the PC.

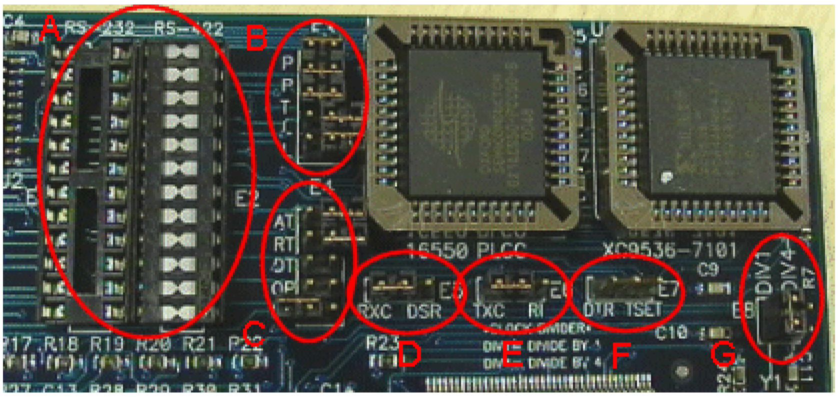
Sealevel PCI ULTRA 530.PCI [7101]

The Sealevel PCI board 7101 requires hardware configuration for the communication with QTM. The jumpers should be set according to the following table, for information on the headers and configuration see the Sealevel manual.

Note: If the board has been bought from Qualisys AB the hardware configuration is already done.

Picture reference	Header on the board	Configuration
A	Header E2	RS422 block selected
B	Header E3	P, P & T selected

C	Header E4	None selected
D	Header E5	RXC selected
E	Header E6	TXC selected
F	Header E7	None selected
G	Header E8	DIV1 selected

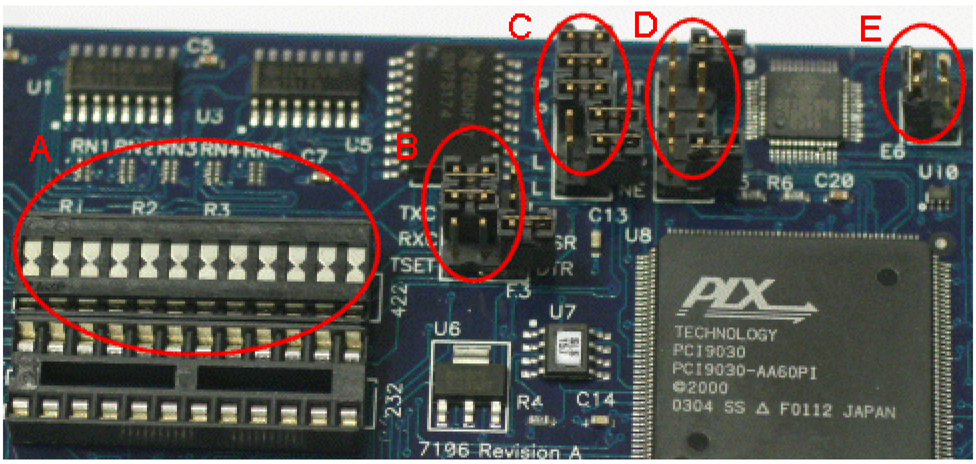


Sealevel Low Profile PCI ULTRA 530.LPCI [7106S-SN]

The Sealevel PCI board 7106 requires hardware configuration for the communication with QTM. The jumpers should be set according to the following table, for information on the headers and configuration see the Sealevel manual.

Note: If the board has been bought from Qualisys AB the hardware configuration is already done.

Picture reference	Header on the board	Configuration
A	Header E2	RS422 block selected
B	Header E3	RXC, TXC selected
C	Header E4	P, P & T selected
D	Header E5	None selected
E	Header E6	D1 selected



Hardware installation PCI and PCI express

Before the board can be installed the power to the PC **MUST** be switched **OFF**, and, to assure extra safety, the power cord removed from the PC.

1. Remove the PC cover.
2. Choose an empty expansion slot. Remove the blanking cover that protects the slot on the PC back panel. KEEP the blanking cover screw for later use.
3. Insert the board in the slot. Make sure that the gold plated PCB fingers fit precisely into the expansion connector. Press firmly but evenly on the top of the board so that it is fitted into the expansion connector. The connections on the board should fit in the slot's aperture on the back of the PC. Use the blanking cover screw to fasten the board.
4. Put the PC cover back on.
5. After attaching all the cables to the PC, switch on the PC.
   If the system does not start as normal, check the following:
   - Make sure that the board is installed correctly.
   - Make sure that the other cards in the PC have not been disturbed.
   - Make sure that the power cord is connected and the PC switched ON!

   If all of these things have been checked and the PC still cannot be switched on, inspect the area surrounding the board to make sure that there are no potentially harmful bits of metal etc. present.

   When the board has been installed it needs drivers to work, for instructions on how to install drivers see chapter “Driver installation Brainboxes (PCI and PCMCIA)” on the facing page, “Driver installation Brainboxes (PCI express and ExpressCard)” on the facing page or “Driver installation Sealevel (PCI and PCMCIA)” on page C - 9.

PCMCIA boards (PC-cards) and ExpressCard

The PCMCIA boards can be installed in laptop computers (Windows XP) with PCMCIA slots. The ExpressCard can be installed in laptop computers (Windows 7) with ExpressCard slots. In the following chapter there is a description of how to install the boards.

Hardware installation PCMCIA and ExpressCard

PCMCIA and ExpressCard boards, by definition, require no hardware configuration and can be installed ”directly from the box”. However for the PCMCIA boards it is important that the computer has a regular COM port for the driver installation to work.

The Brainboxes ExpressCard is sold together with a extension so that it fits better in the wider types of ExpressCard slots.

The boards are ‘hot plug’ compatible and it may therefore be inserted into the PCMCIA type II respectively ExpressCard slot when the machine is either switched off or on. Check the computer user manual for detailed instructions on how to insert the cards in the slots.

When the board has been installed it needs drivers to work, for instructions on how to install drivers see chapter “Driver installation Brainboxes (PCI and PCMCIA)” on the facing page, “Driver installation Brainboxes (PCI express and ExpressCard)” on the facing page or “Driver installation Sealevel (PCI and PCMCIA)” on page C - 9.

Driver installation Brainboxes (PCI express and ExpressCard)

The following procedure is used to install the drivers of Brainboxes PCI express and ExpressCard on Windows 7.

The drivers for these boards are located automatically by Windows update. Just select to install them when Windows gives you the message after the boards have been installed.

If the computer has no internet connection there are drivers included with the QTM installation. The drivers are located in the **Brainboxes (Windows 7)** folder in the **Drivers** folder in the **Qualisys Track Manager** folder (usually C:\Program files (x86)\Qualisys\Qualisys Track Manager\Drivers\Brainboxes). Unzip the drivers for the respective board; Boost.PCIe_v3.0.17.0.zip for PCI express (desktop) and Boost.Software_2.0.78.zip for the Expresscard (laptop). Then in the Windows driver installer browse to the location of the drivers.

Driver installation Brainboxes (PCI and PCMCIA)

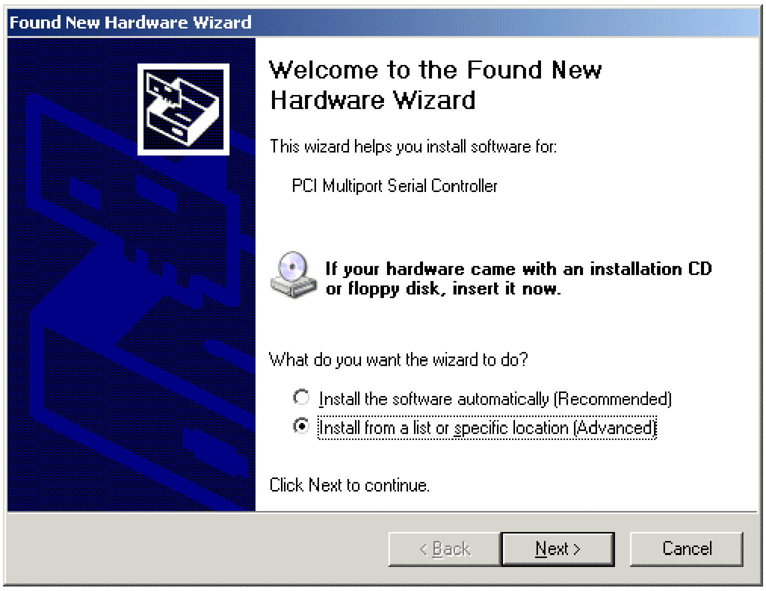
The following procedure is used to install the drivers of Brainboxes PCI and PCMCIA boards on Windows XP.

The drivers of the Brainboxes boards are located in the **Brainboxes** in the **Drivers** folder in the **Qualisys Track Manager** folder (usually C:\Program files\Qualisys\Qualisys Track Manager\Drivers\Brainboxes). In the following instructions the driver version is 5.4.52, however the instructions can be applied to any version of the drivers.

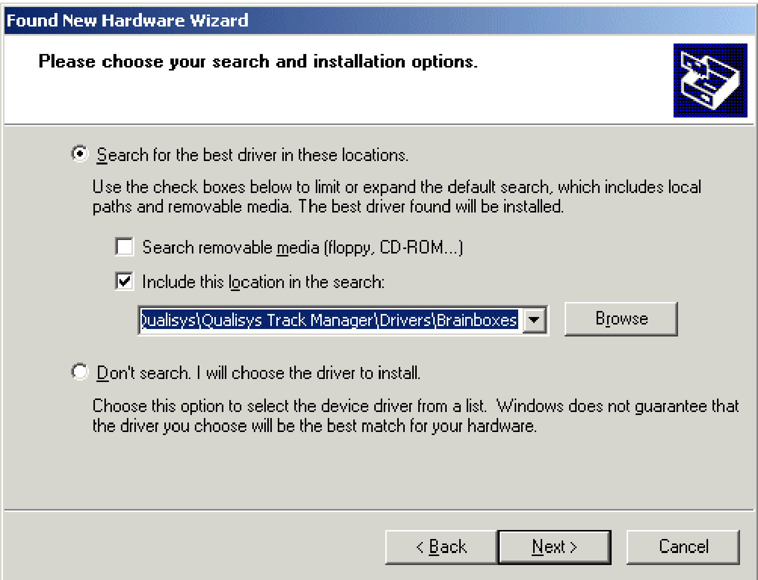
Note: For Brainboxes PCMCIA on new computers you must use a later driver then the one supplied by Qualisys. Contact Qualisys AB if you need that driver.

1. The drivers are either installed when the computer is started after the board has been installed or after Windows has started with the **Add Hardware** wizard on the **Control panel**. The following dialogs will appear in both cases.

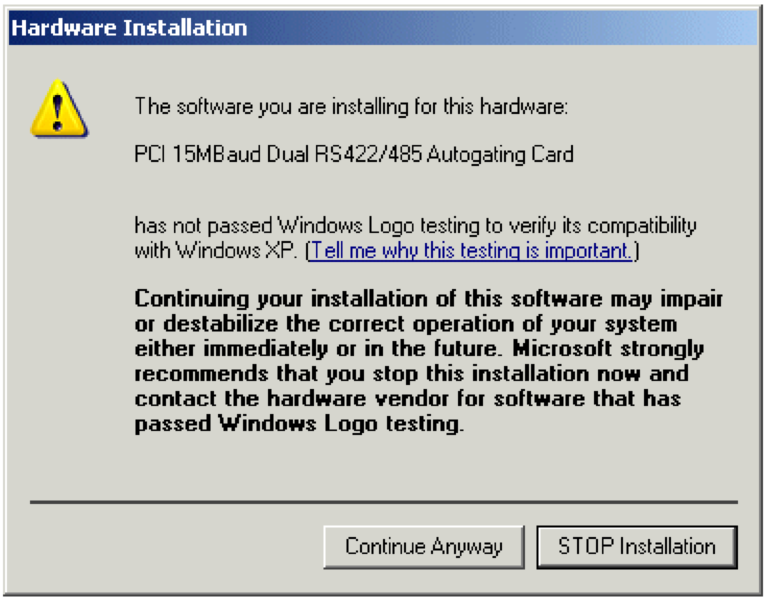
Click **Next**.



The wizard has found the PCI board or if it is a laptop the PCMCIA board. Select **Install from a list or specific location** and click **Next**.

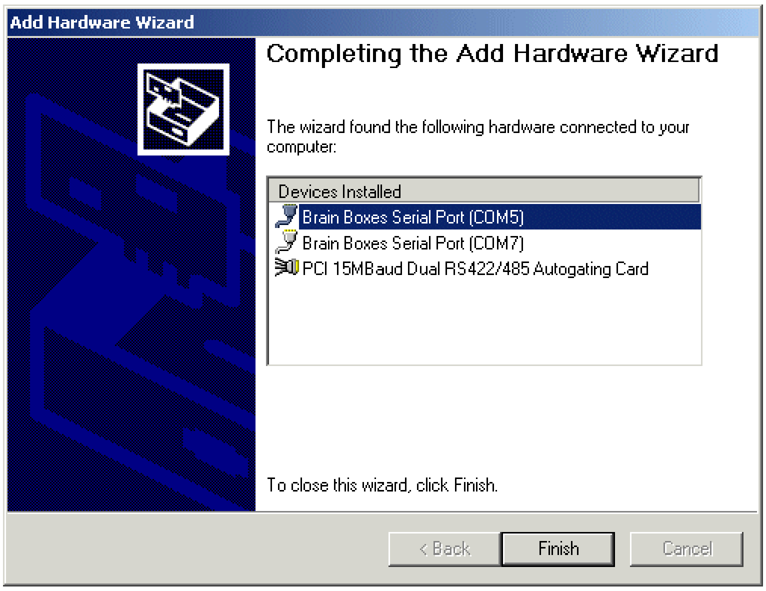


Select **Search for the best driver in these locations** and **Include this location in the search**. Then click **Browse** and locate the **Brainboxes** folder with drivers in the **Drivers** folder.



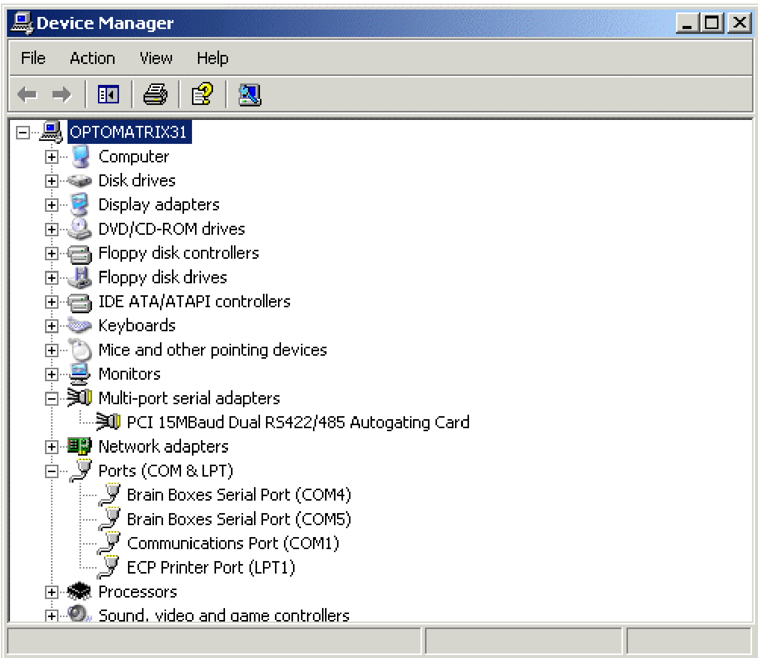
Click **Continue Anyway**.

Click **Finish**. Sometimes the wizard will install the ports automatically. Otherwise the same dialogs as for the board will appear for the ports and you can just follow the instructions for the dialogs above.



This dialog shows the updated hardware. Check that it is the correct board, i.e. **PCI 15MBaud Dual RS422/485 Autogating Card** and two **Brain Boxes Serial Ports** or for a laptop one **Brain Boxes Limited – 1 Port PCMCIA 422/485 Card** and one **Brain Boxes Serial Port**. Click **Finish**.

2. After the update it is important to check that drivers are ok, but first you should restart the computer. When the computer has started again open the **Device Manager**.



Check that the board exists under the **Multi-port serial adapters** branch, for a PCI board it should be called **PCI 15MBaud Dual RS422/485 Autogating Card**. For the PCI board there should also be two **Brain Boxes Serial Ports** under the **Ports (COM & LPT)** branch.

For a PCMCIA board there should be a **Brain Boxes Limited – 1 Port PCMCIA 422/485 Card** under the **Multi-port serial adapters** branch and just one **Brain Boxes Serial Port** under the **Ports (COM & LPT)** branch.

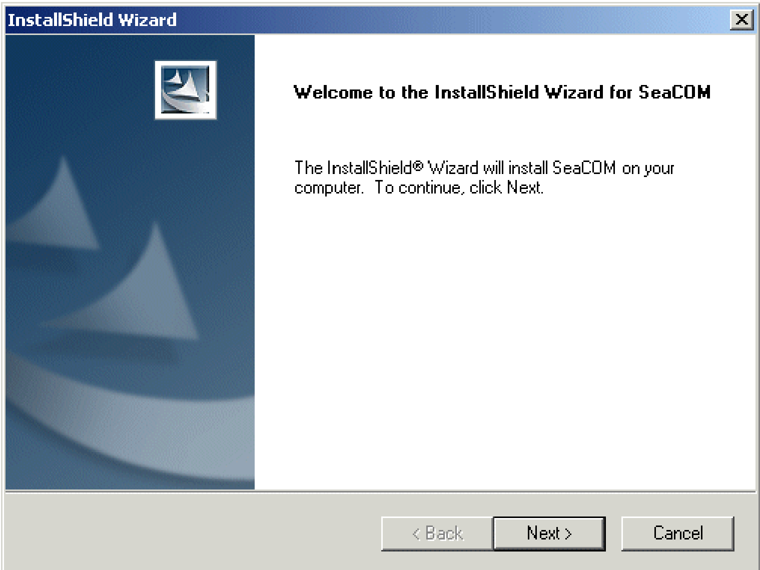
Also make sure that the board and its ports are running ok, i.e. that there is no exclamation mark in front of the unit.

Driver installation Sealevel (PCI and PCMCIA)

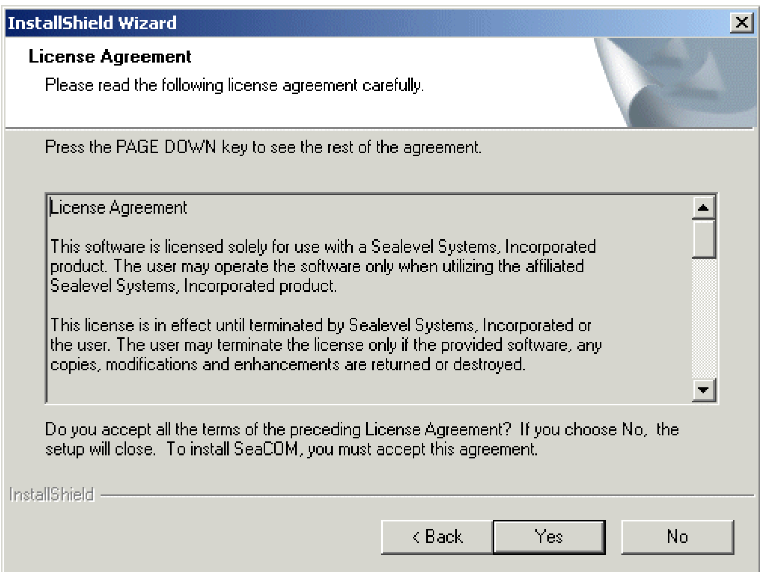
The installation of drivers for Sealevel boards is described for Windows XP. The program which upgrades the drivers for the Sealevel boards are located in the **Sealevel** folder in the **Drivers** folder in the **Qualisys Track Manager** folder (usually C:\Program files\Qualisys\Qualisys Track Manager\Drivers\Sealevel). In the following examples the driver version is 2.5.29, however the instruction can be applied to any version of the drivers.

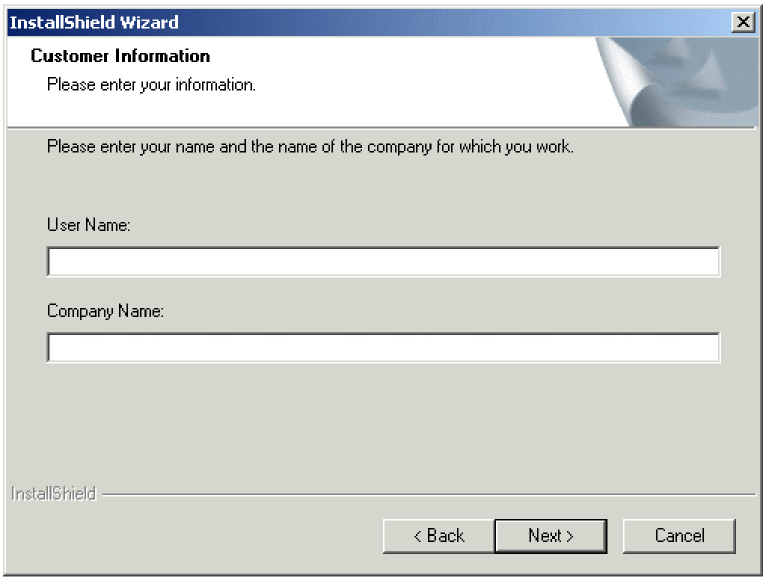
The following procedure is used to install drivers for Sealevel PCI and PCMCIA boards on Windows XP.

1. The drivers for the Sealevel board are installed with a special program. For the current version it is called SS020529.exe. This program must be run when Windows has started and therefore the **Add hardware** wizard should be cancelled if it appears during the startup. Run the program SS020529.exe.

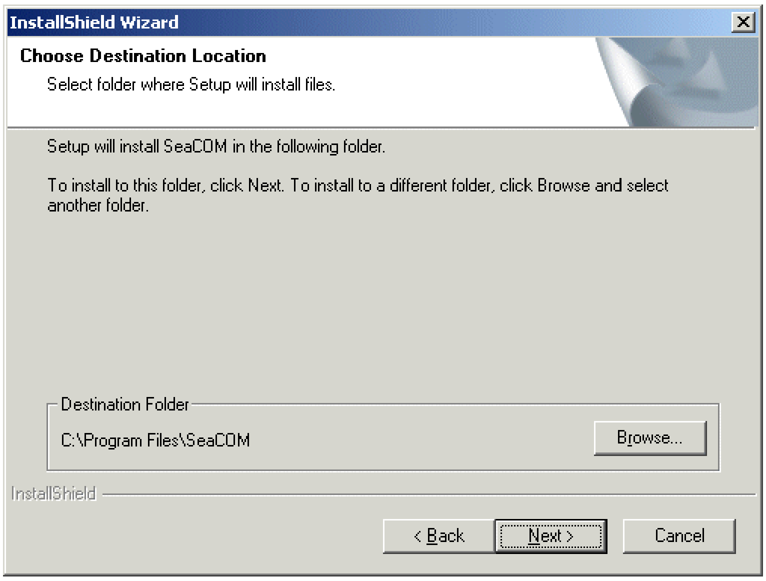


Click **Next**.

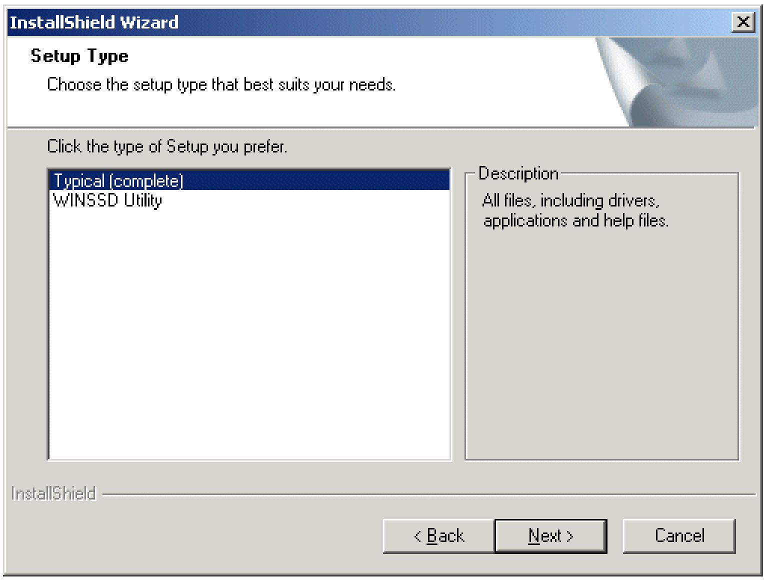




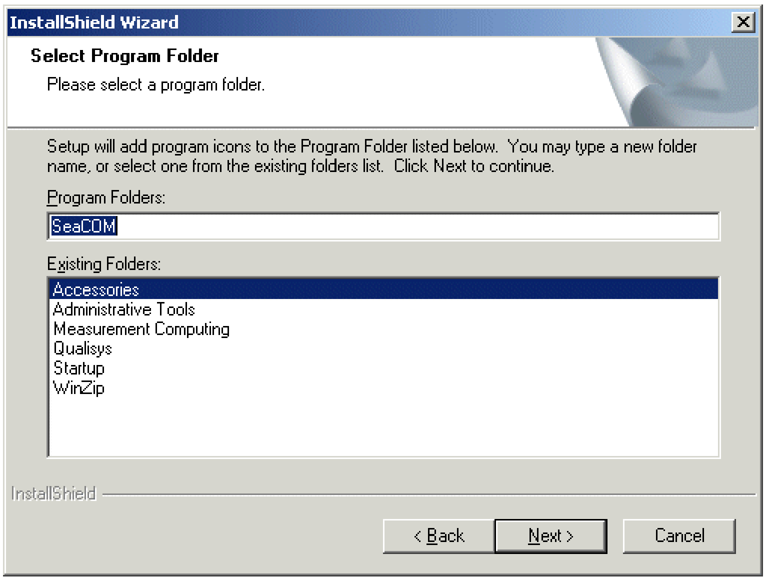
Write your name and company name. Click **Next**.



Click **Next**.

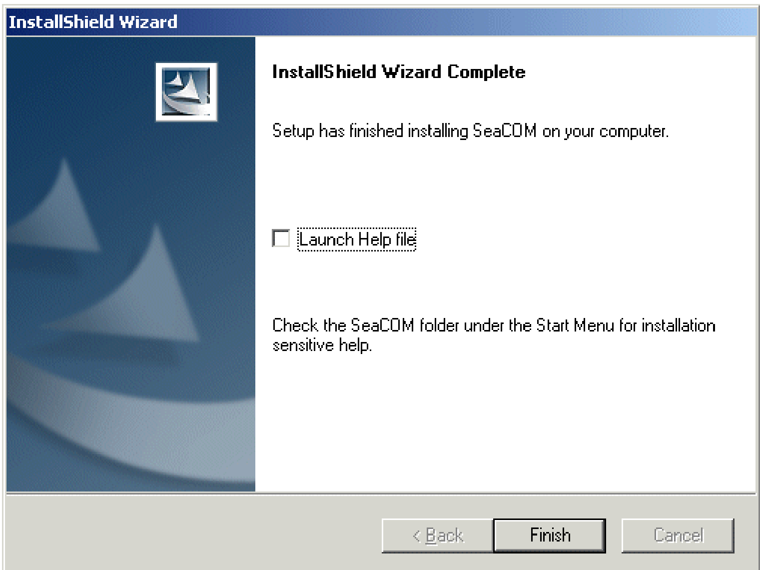


Select **Typical (complete)** and click **Next**.



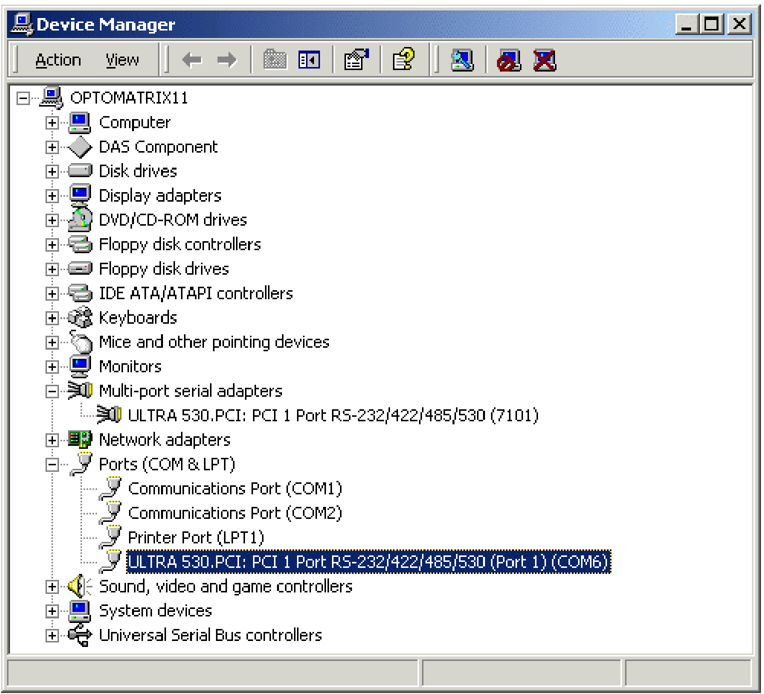
Click **Next**.

Click **OK**.



Click **Finish**. The drivers have been installed.

2. After the update it is important to check that drivers are ok, but first you should restart the computer. When the computer has started again open the **Device Manager**.

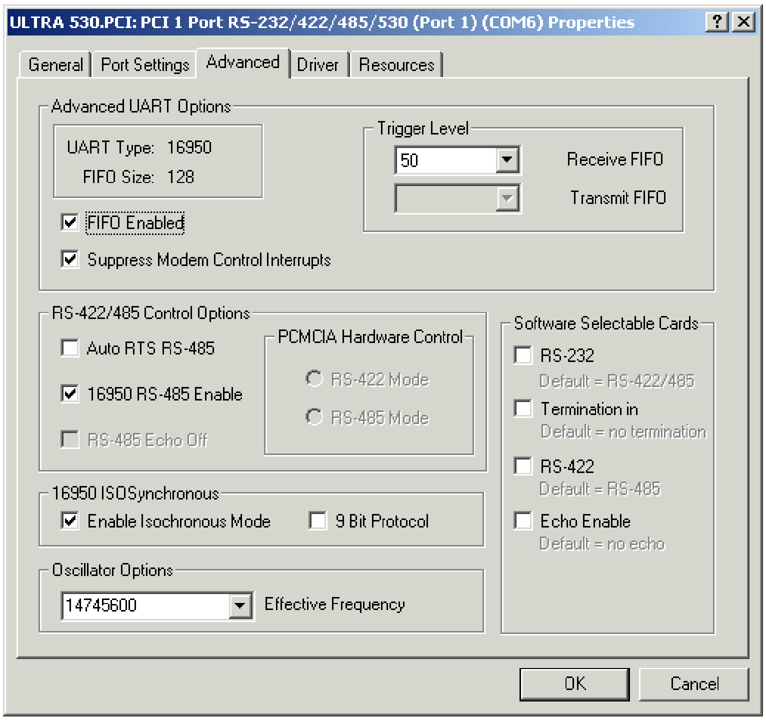


For PCI boards, check that the board exists under the **Multi-port serial adapters** branch, it should be called **ULTRA 530.PCI: PCI 1 Port RS-232/422/485/530 (7101)**. There should also be one **ULTRA 530.PCI: PCI 1 Port RS-232/422/485/530 (Port1)** port under the **Ports (COM & LPT)** branch.

For PCMCIA boards there is just one port called **ULTRA PC-SIO: PCMCIA RS-422/485 Serial Port (3602)** under the **Ports (COM & LPT)** branch and nothing under the **Multi-port serial adapters** branch.

Also make sure that the board and its ports are running ok, i.e. that there is no exclamation mark in front of the unit.

Finally you should set the settings of the board. This is done on the properties of the port. Double-click on the port (under the **Ports (COM & LPT)** branch) to open the **Properties** dialog. Then click on the **Advanced** tab and set the settings according to the screen dump below.



Note: If it is a PCMCIA board the **PCMCIA Hardware Control** setting should be set to **RS-485 Mode**.

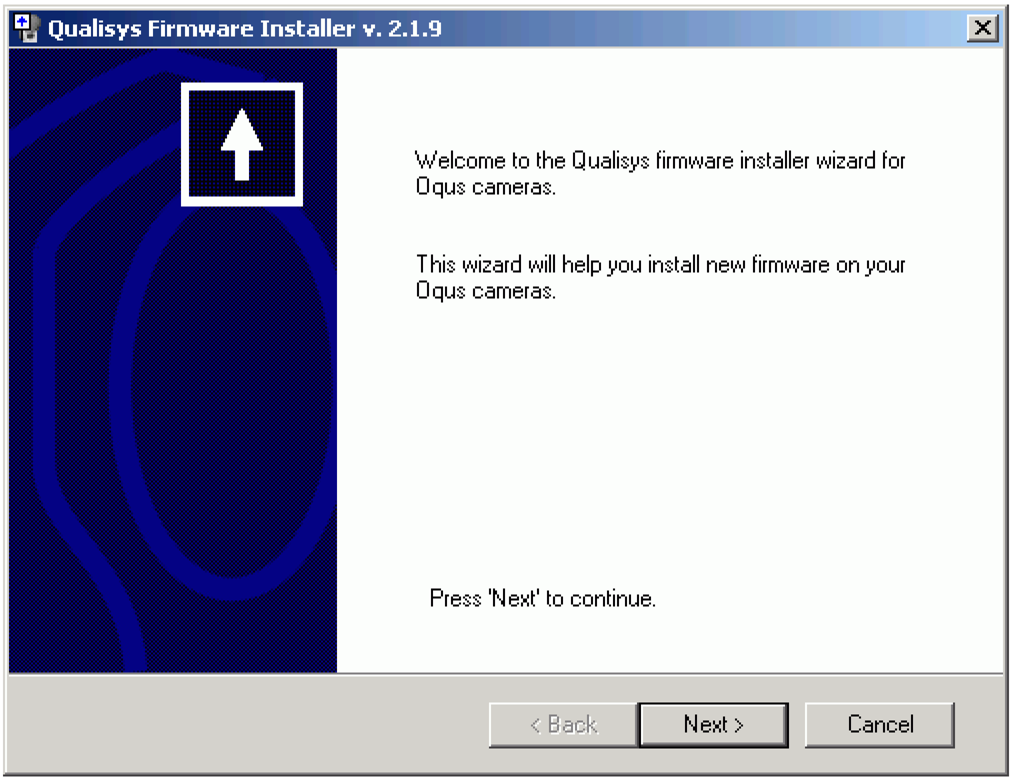
Appendix D: Qualisys Firmware Installer (QFI)

QFI.exe

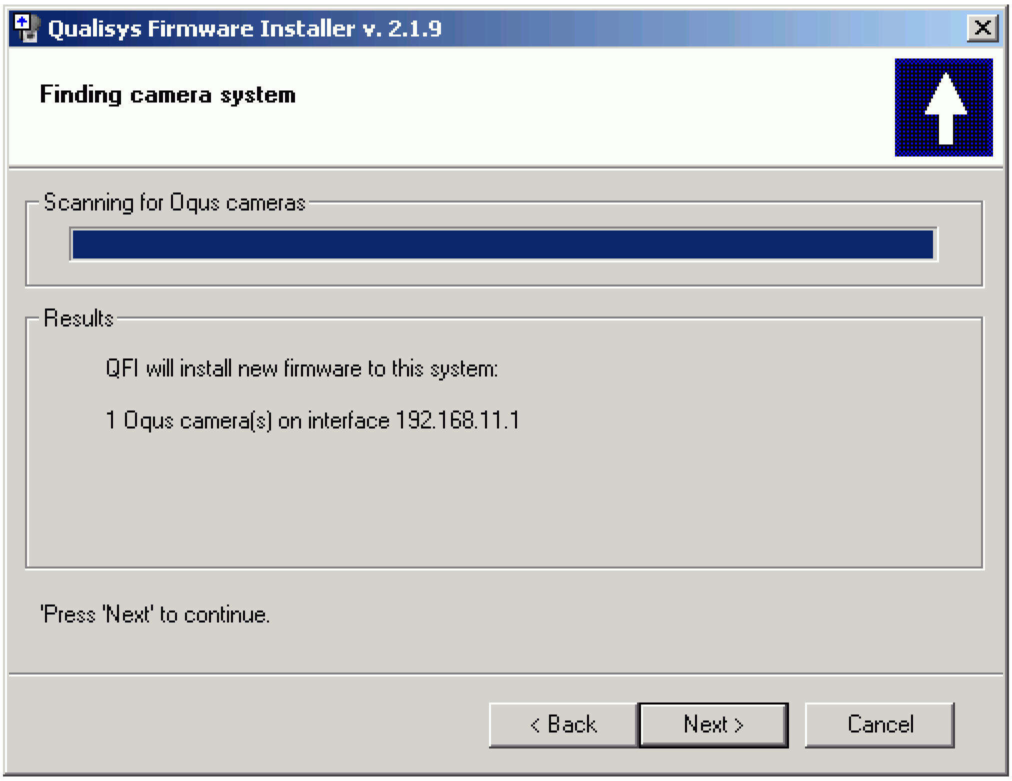
The QFI (Qualisys Firmware Installer) is used to download firmware to cameras. The program has different GUIs for Oqus and ProReflex, see chapters “QFI (Oqus)” below and “QFI (ProReflex)” on page D - 4.

QFI (Oqus)

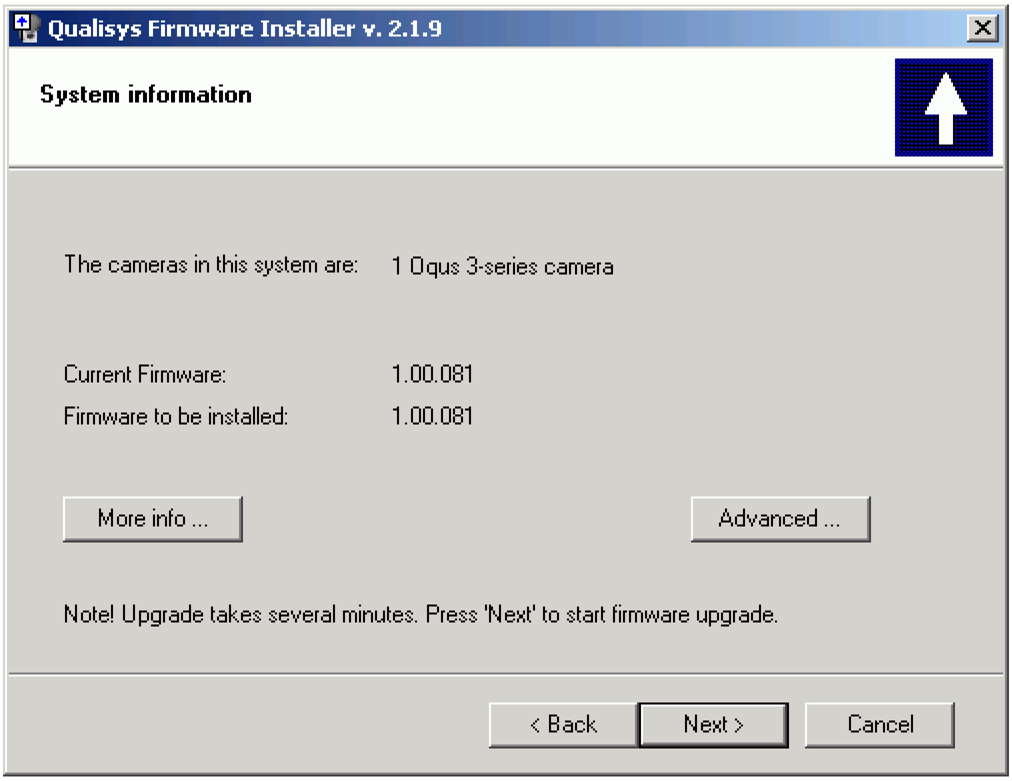
The QFI for Oqus is organized as a Wizard. In a regular firmware upgrade just click **Next** until the upgrade is finished, an upgrade will take a few minutes to perform. The different steps are described below.



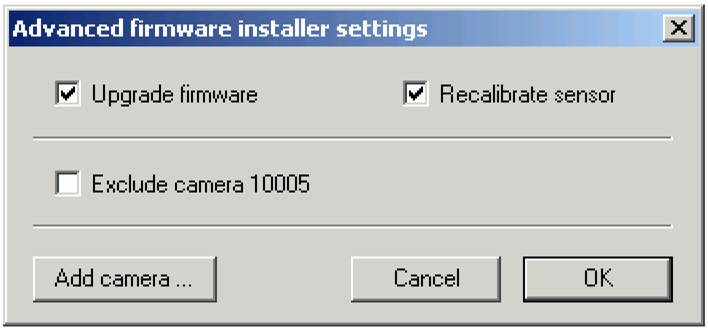
Click **Next** to start looking for Oqus cameras connected to the computer. The cameras must have started completely before you start locating the cameras.



Click **Next**.



Look at the camera system that has been located and that the **Firmware to be installed** is correct. Click on **More info** for more information about the cameras. Click on **Advanced** to open the **Advanced Firmware installer settings** dialog below. Click **Next** to start the firmware installation.

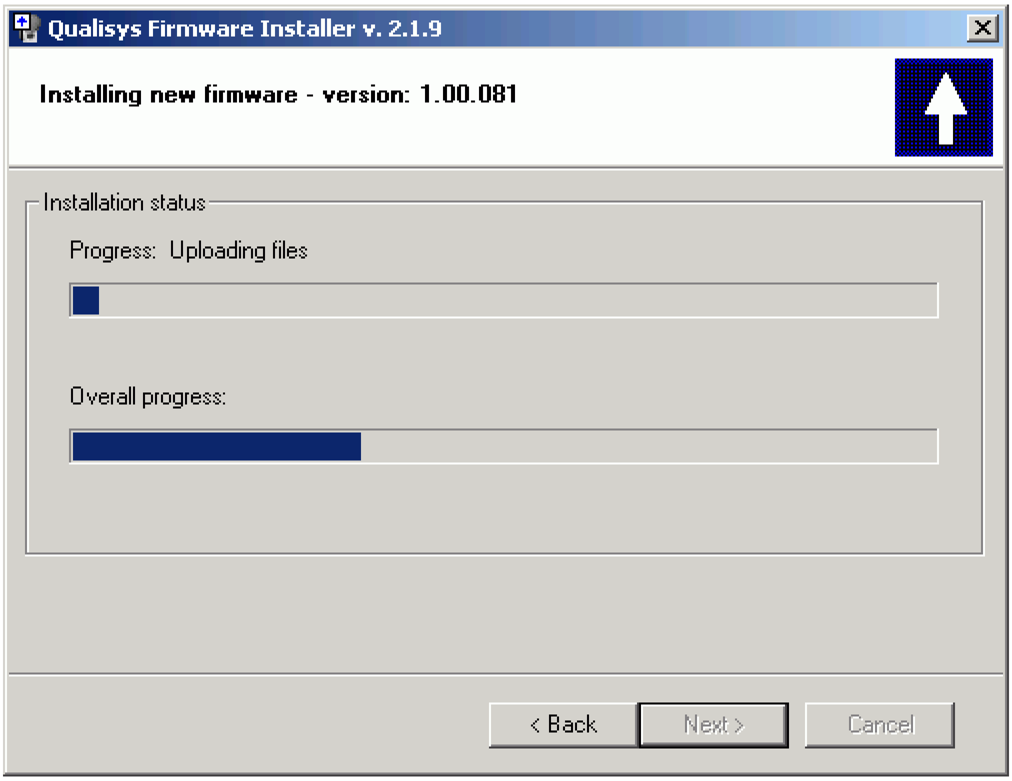


The **Advanced Firmware installer settings** dialog contains settings that can exclude some parts of the firmware installation. Do NOT use these settings unless you are absolutely sure.

Upgrade firmware
Uncheck to not download the new firmware, which is useful if you only need to **Recalibrate sensor**.

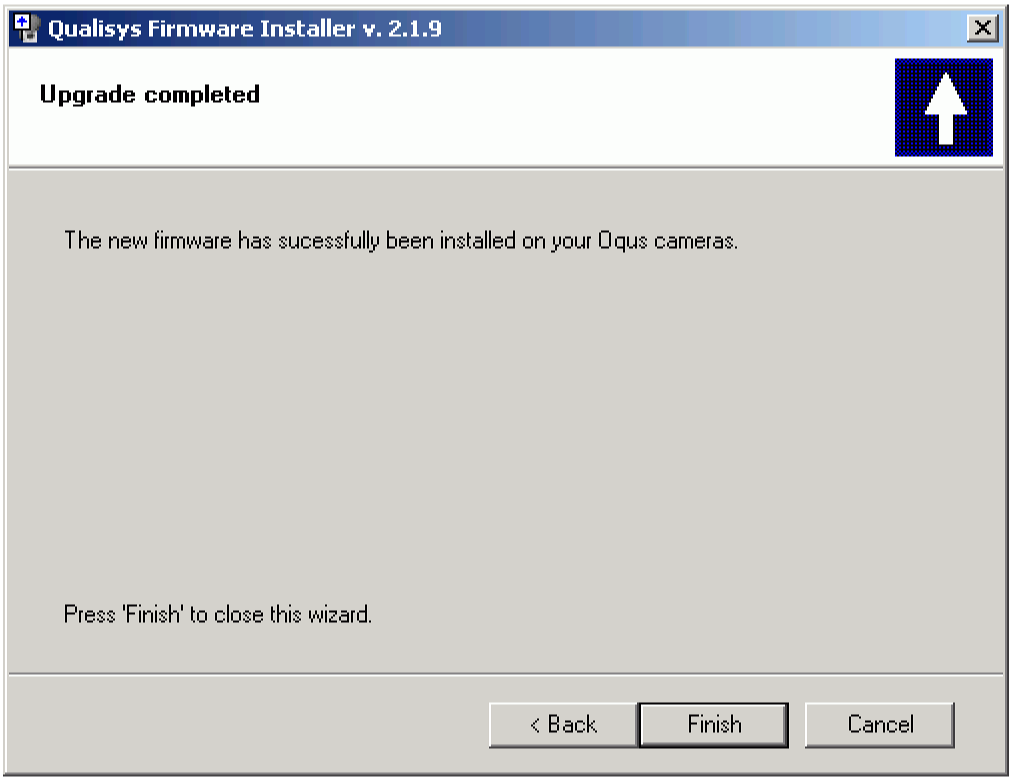
Recalibrate sensor
Uncheck to not recalibrate sensor, the recalibration is only needed for Oqus 3- and 5-series. Do NOT uncheck this setting unless you are absolutely sure that the sensor is already calibrated for this firmware.

Below these settings are a list of all cameras in the system. Check the cameras that you do not want to upgrade.



Wait until all of the four steps below have finished. The progress will sometimes jump back, but that is not a problem.

1. Uploading files
2. Programming camera(s)
3. Waiting for camera(s) to reboot
4. Camera sensor calibration



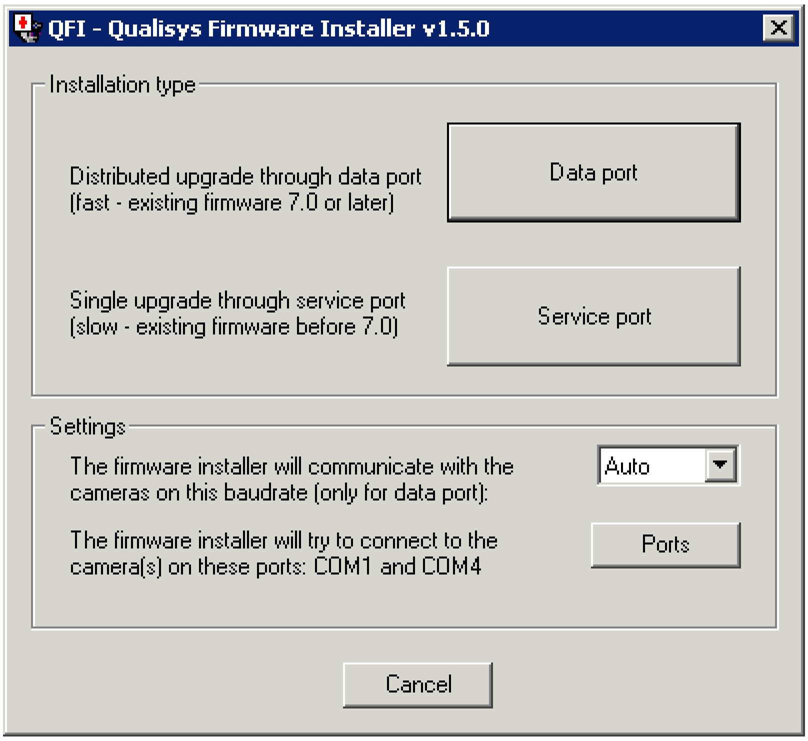
Click **Finish**.

QFI (ProReflex)

The program QFI.exe is used to upgrade the firmware of a ProReflex MCU. It can also be used to just change the ID of the camera or to retrieve some camera information. To download a certain firmware open that folder in the **MCU_firmware** folder in the **Qualisys Track Manager** folder and use the QFI.exe that is located in that folder.

The connection to the MCU depends on the current firmware version in the MCU. For versions before 7.0 a RS232 port on the measurement computer is connected to the **Service** port of the MCU. For versions 7.0 and later the distribution can be done directly on the **Data** port, i.e. the same connection can be used as for measurements. This is much faster and makes it possible to upgrade all the MCUs in the system at once.

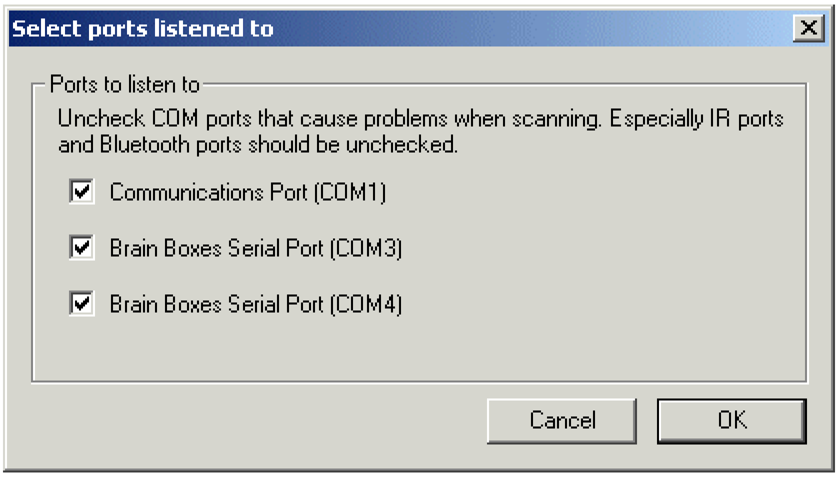
Start the program (QFI.exe) and choose whether the connection is to the **Data** port or to the **Service** port, for descriptions on how to upgrade the firmware see chapters below.



Under the **Settings** heading you can select the baud rate for the **Data** port and which ports the installer will use when looking for the MCU system. These settings should only be used if the automatic detection fails.

The baud rate is changed on the drop-down list. The default setting is **Auto**, which means that the baud rate is set to the maximum value of the current COM port. Otherwise the baud rate can be set manually to a value between 9600 and the maximum of the current COM port.

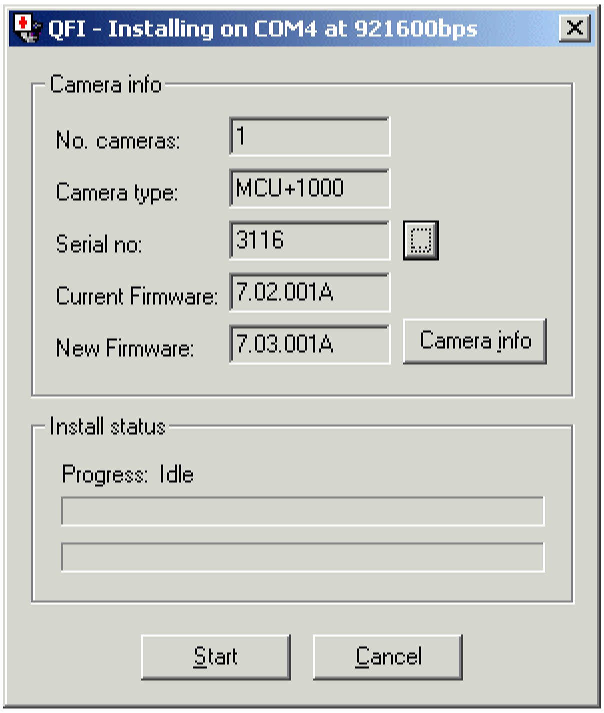
To select the port click on **Select Ports**, this will open the dialog below.



In the dialog above you can select the COM ports that QFI will use when looking for the MCU system. This can help if QFI cannot find the system, especially IR and Bluetooth ports can cause problems and should therefore be deselected.

Firmware update via the data port

The data port can be used if the firmware in the MCU is 7.0 or later. When upgrading via the data port the camera system can still be connected as for measuring. The firmware can be updated as soon as QFI has connected to the system and the dialog below is displayed.



The QFI main window shows the following information of the MCU system:

The current COM port and the baud rate are shown in the title bar.

No. cameras
The number of cameras in the system.

Camera type
The type of camera.

Serial no
The serial number of the cameras in the system.

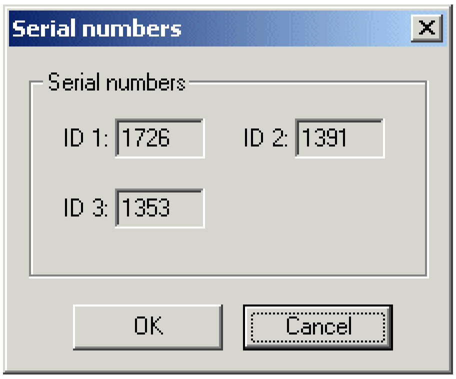
Current firmware
The firmware currently used by the camera.

New firmware
The new firmware that will be installed in the camera.

If the new firmware is correct click **Start** to install the firmware. The progress of the installation will be shown under the **Install status** heading.

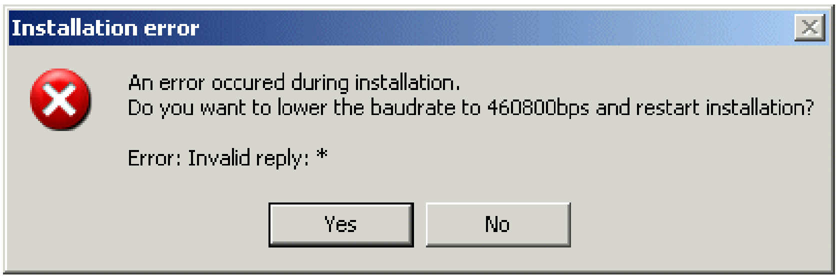
Note: All of the cameras in the system will be updated at the same time.

When there are more than two cameras in the system the **Serial no.** will not fit in the display. Then you can click on the little box next to the display to open a dialog with all of the serial numbers.



With the **Camera info** option a dialog is shown with information about the hardware and the firmware in the cameras.

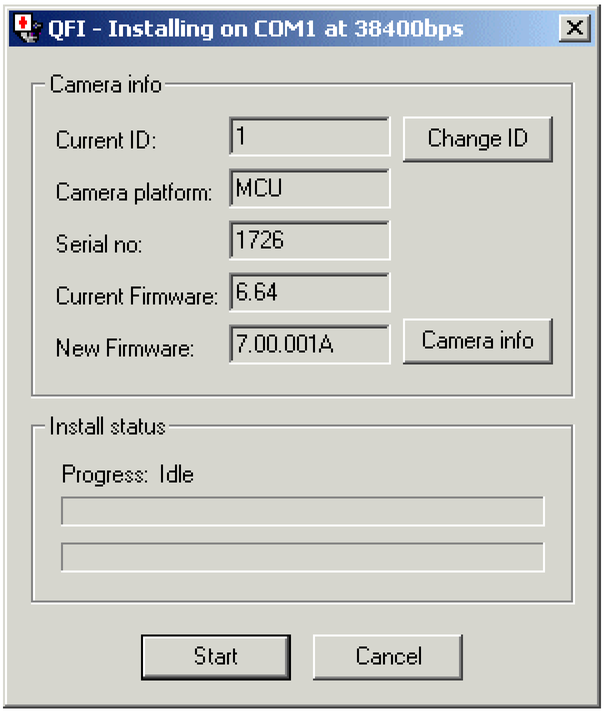
If there is any error during the firmware installation, QFI will show the dialog below where you can choose to restart the installation automatically at a lower baud rate.



Firmware update via the service port

The service port must be used when the firmware in the MCU is earlier than 7.0. When updating via the service port just one MCU at a time can be connected to QFI. Follow these steps to connect QFI to a ProReflex MCU via the service cable:

1. Disconnect all of the cables from the MCU except the power cord.
2. Connect the service cable, which was delivered with the ProReflex cameras, to the serial communication port of the PC (usually COM1 or COM2), **NOT** the Brainboxes board.
3. Connect the other end of the Service cable to the **Service** port of the ProReflex camera.
4. Start QFI.exe and click **Service port**.
5. Start the camera, it must be restarted and it might take some time before the QFI main window is shown. During that time a dialog is shown with the **Port status** and the program can be shut down with **Cancel**.



The QFI main window shows the current information of the MCU.

Current ID
The ID which is used to identify the camera in the system.

Camera platform
The type of camera.

Serial no
The serial number of the camera.

Current firmware
The firmware currently used by the camera.

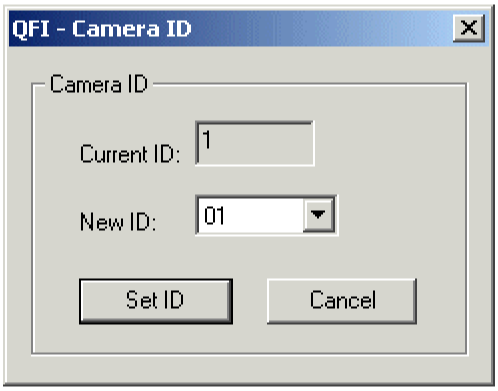
New firmware
The new firmware that will be installed in the camera.

If the new firmware is correct click **Start** to install the firmware. The progress of the installation will be shown in the **Install status**.

Important: Every ProReflex camera in the camera system must be upgraded.

The ID of a camera can be changed by clicking **Change ID** to open the dialog below. Choose the **New ID** and click **Set ID** to change the ID. If the firmware should not be updated you can click on **Cancel** in the QFI main window.

Note: Make sure that the ID is not used by another camera in the system.



With the **Camera info** option a dialog is shown with information about the hardware and the firmware in the MCU.

Appendix E: Analog capture hardware

Analog capture hardware supported by QTM

To use the QTM software for synchronous motion and analog voltage data retrieval, the following additional components are required:

1. A Measurement Computing Inc (formerly Computerboards) A/D board with supporting drivers, driver version 6.1 or higher.
   QTM supports the following analog boards:
   - USB-2533
   - USB-1616FS
     Note: Does only work well on old computers using Windows XP.
   - PCI-DAS6402/16
     Note: If you are to update the computer contact Qualisys AB for a computer chassis where the board fits.
   - See also chapter “Old analog hardware” on next page.
2. A synchronization cable.
3. A BNC connection box for the PCI and PC-CARD.

Normally the entire system is installed in the measurement computer before it is delivered by Qualisys AB. However, if you have an USB A/D board or if you are upgrading the system to use analog signal acquisition with a PCI board, it is important to follow the instructions in the following chapters.

- For an USB A/D board read the chapters:

  “Installing drivers for the A/D board” on next page

  “Installing the USB-2533 board” on page E - 6 or “Installing the USB-1616FS board” on page E - 9

  “USB A/D board” on page E - 4

  “Hardware synchronization” on page E - 13

- To upgrade the system to use analog signal acquisition with a PCI board read the following chapters:

  “Installing drivers for the A/D board” on next page

  “Installing the PCI A/D board” on page E - 4

"Hardware synchronization" on page E - 13

Old analog hardware

The following analog hardware is no longer tested in QTM, they will probably continue working on old computers, but it cannot be guaranteed.

PCI-DAS1602/16
The PCI-DAS1602/16 board will soon be discontinued from Measurement computing and the drivers are no longer updated. With the current drivers the board will not trigger on new computers, even if they use Windows XP. This means that the board only works on old computers using Windows XP. Because there will be no solution to this problem we have decided to stop testing the board in QTM 2.6.

PC-CARD DAS16/16
The PC-CARD DAS16/16 is no longer tested in QTM, because the PCMCIA slots are being removed from new laptops and it will therefore soon be difficult to use.

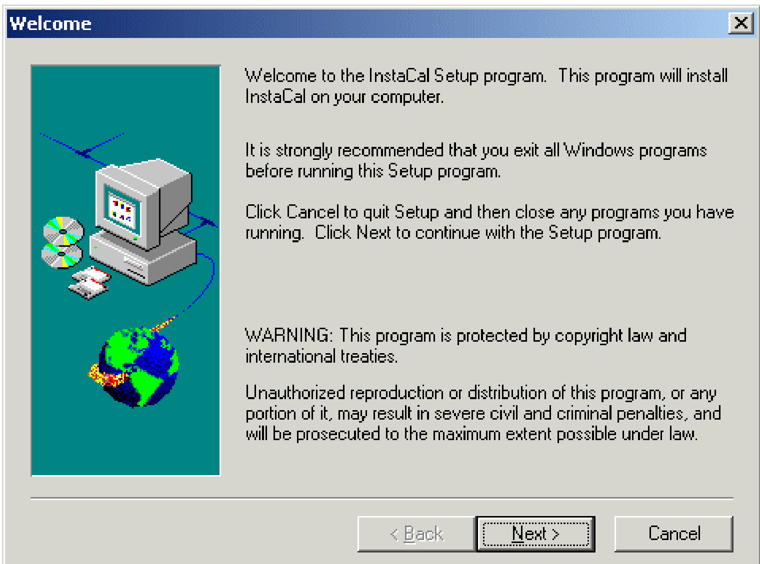
PCIM-DAS1602/16
The PCIM-DAS1602/16 are no longer tested in QTM, because it is an old board and there was only a handful sold by Qualisys. If you have this board contact Qualisys AB.

Installing drivers for the A/D board

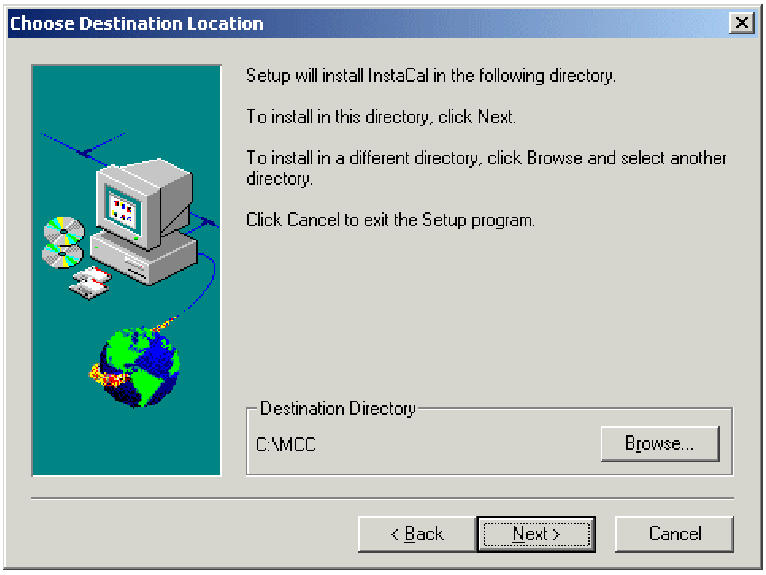
The Instacal software (drivers for the A/D board) must be installed on the computer before the installation of the hardware. The installer is included on a CD with the analog board. If you do not have the CD the latest version can always be downloaded from www.mccdaq.com.

Note: Instacal version 6.1 or later must be used on Windows 7.

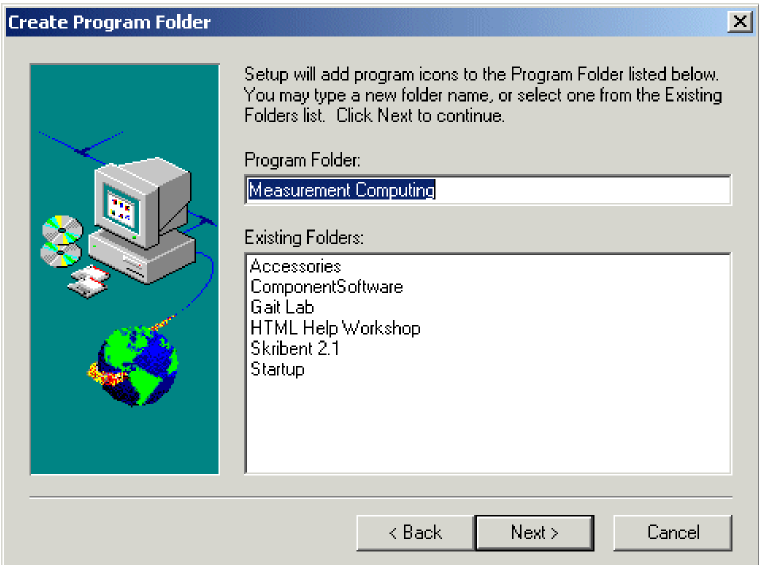
To install the software, run the **Setup.exe** file and follow the instructions below.



Click on **Next**.



Click on **Next**.



Click on **Next**.

Click on **No**.

Select **Yes, I want to restart my computer now** and then click on **OK**.

When the computer has been restarted you can go on to install your A/D board, see chapter “Installing the PCI A/D board” below, “Installing the USB-2533 board” on page E - 6 or “Installing the USB-1616FS board” on page E - 9. The PCI board, PC-Card or USB board is then configured via the Instacal software.

For more information about InstaCal read the **README.TXT** file from Measurement Computing that is put in the folder for the Instacal software.

Installing the PCI A/D board

Turn off the computer and remove the power cord. Insert the A/D board in the computer and switch on the computer. Run the Instacal software on the **Windows Start** menu and choose the correct board:

16 **Single Ended** channels (PCI-DAS 1602/16)

64 **Single Ended** channels (PCI-DAS 6402/16)

16 **Single Ended** channels (PCIM-DAS 1602/16)

16 **Single Ended** channels (PC-Card 16/16)

Open the **Properties** dialog for the board and change the **No of channels** to **xx Single ended**. The **Differential** setting does not work because the BNC connection box is wired for single-ended measurements.

When the board is properly installed it will be listed on the **Analog boards** page in the **Project options** dialog in QTM.

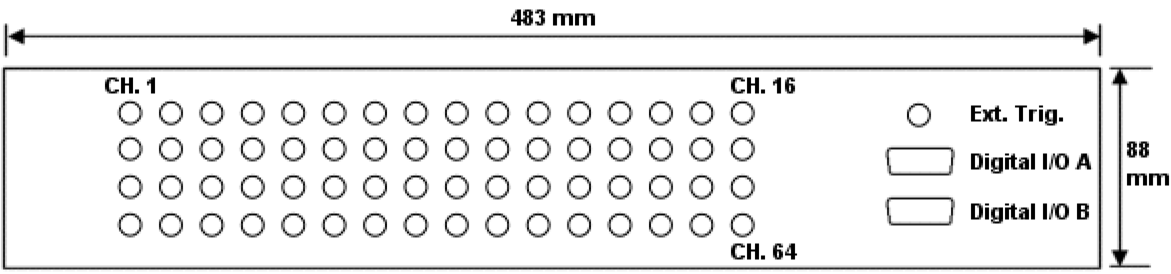
Note: There are no hardware settings available on the A/D boards (except for the ”PCIM-DAS1602/16” board).

USB A/D board

Qualisys supports the 64 channel board USB-2533 and the 16 channel board USB-1616FS.

USB-2533

The USB A/D board (USB-2533) is a portable A/D board that can easily be connected to any computer with an USB port. The board has 64 analog channels and is distributed in a case with BNC connections for the analog signals and the synchronization signal.



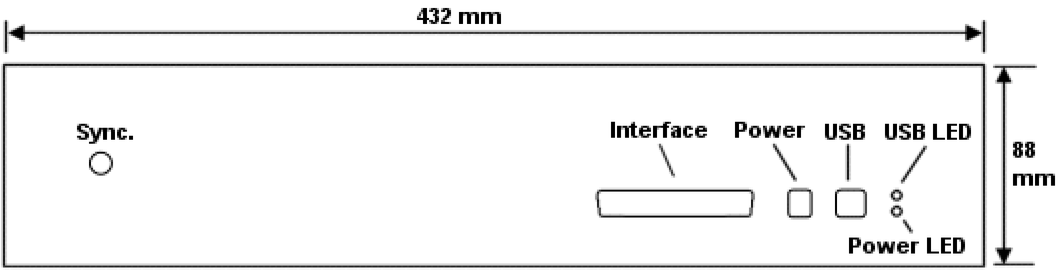
The following connections are available on the front view of the board:

CH. 1 - CH. 64
BNC connections for the 64 analog channels.

DIGITAL I/O A and **DIGITAL I/O B**
Ports for controlling a Kistler force plate, the cable can be ordered from Qualisys.

Note: The port can also be used to control other applications. Pin 1-12 on each port is then the digital I/O and pin 13-16 is ground.

EXTERNAL TRIGGER
BNC connection for the synchronization cable, see chapter “Hardware synchronization” on page E - 13.



The following connections are available on the rear view of the board:

SYNC
BNC connection for synchronization of several USB A/D boards. Not implemented in QTM yet.

INTERFACE

WARNING:Do not use this connection. If there are connections both on the front and on this interface then the board can be damaged.

POWER
Connection for the power supply, which is supplied by Qualisys.

USB
USB connection to the measurement computer. The cable is supplied by Qualisys.

USB LED
The USB LED is lit when the board is connected to a computer.

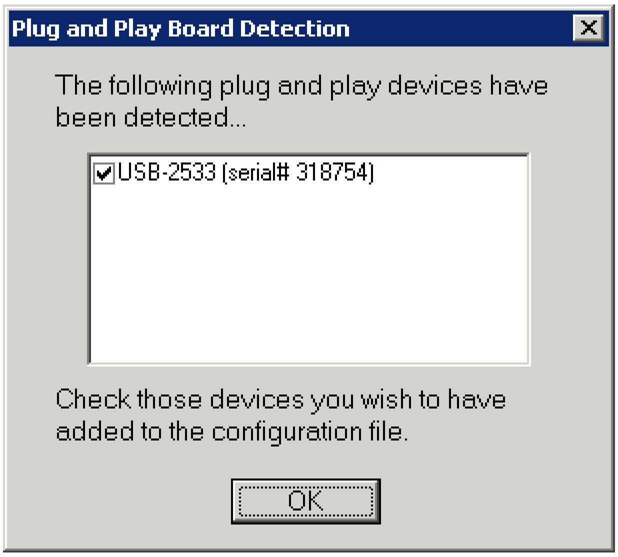
POWER LED
The Power LED is lit when the board has power. The power can come either from the USB port or from an external power supply. To use the external power supply it must be plugged in before the USB connection, however the Power LED will still not be lit until the USB cable is connected.

Installing the USB-2533 board

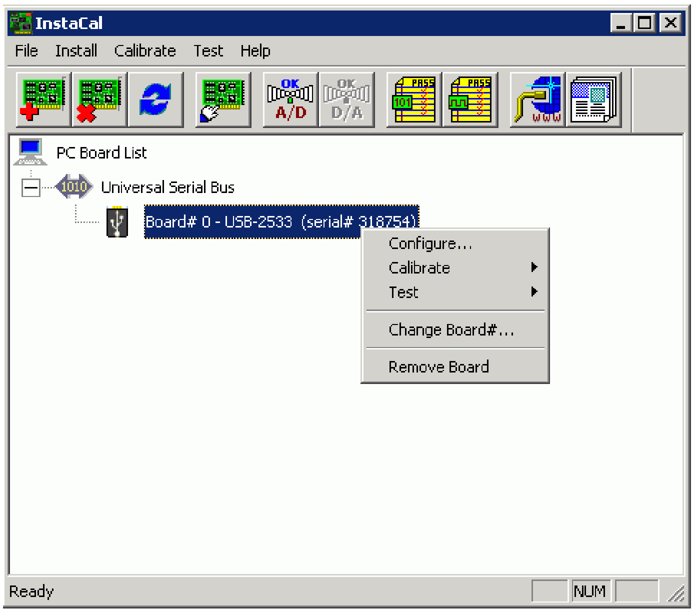
The computer must have Windows XP Service pack 2 or a later Windows version for the board installation to work.

Follow this procedure to install the USB A/D board:

1. Connect the supplied power supply to the **POWER** connection. It is important to connect the power supply before the USB, because when power adapter is connected to the board it cannot get the power from the USB connector. With a laptop the power supply must always be used otherwise the analog board will not start.
2. Connect the **USB** port on the analog board to an USB port on the measurement computer.
3. The **Found New Hardware Wizard** will be opened twice the first time you install a USB-2533 board on a computer. First it installs the MCC USB2 Loader Device and then it installs USB-2533 board. In the installation use the options **No, not this time** and **Install the software automatically (Recommended)**.
4. Restart the computer.
5. Run the Instacal software on the **Windows Start** menu.
6. Instacal will detect the USB A/D board and display the following dialog.



7. Click **OK**. The board will then be listed in Instacal as shown below.



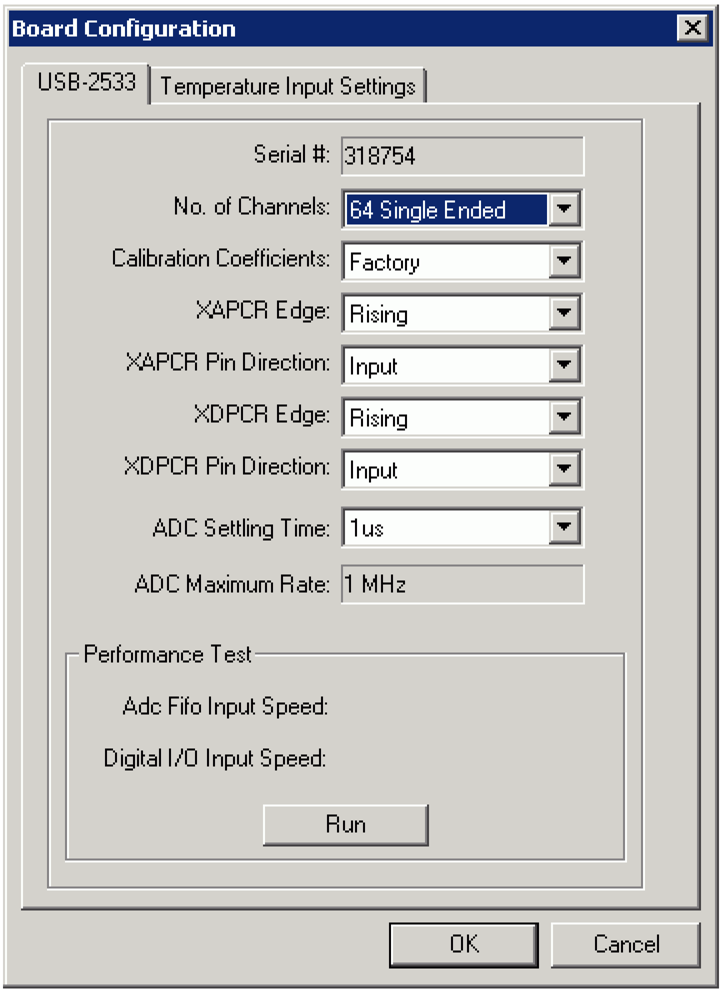
8. Right-click on the board and then click on **Configure**. Then change **No. of Channels** to **64 Single Ended**. The reason for using single ended is among other things that force plates usually have this type of signal output.

   Note: It is possible to run this board in differential mode. However the wiring inside the connection box has not been optimized for differential signals. For differential signals HI and Lo is separated by 8 channels, e.g. Channel 1 HI and LO would be connected to Ch. 1 respectively Ch. 9.

   With Oqus cameras and the **Use external sync** option for the analog board in QTM it is important to check that the **XAPCR Edge** setting so that it matches the setting for the camera sync output on the **Advanced (Timing)** page in the **Project options** dialog. The default **TTL signal polarity** is **Negative** which matches **XAPCR Edge** = **Falling**, which then means that **Positive** equals **Rising**.

   Note: For ProReflex it is best to keep the **XAPCR Edge** as **Rising**.

   Warning: Do not change the **XAPCR Pin Direction** setting to **Output** as it may damage the board or the camera.

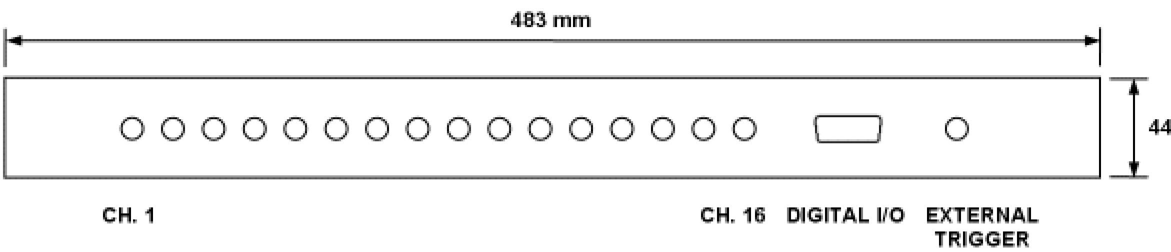


9. Click **OK** and then exit Instacal.

When the board is properly installed it will be listed on the **Analog boards** page in the **Project options** dialog in QTM.

USB-1616FS

The USB A/D board (USB-1616FS) is a portable A/D board that can easily be connected to any computer with an USB port. The board has 16 analog channels and is distributed in a case with BNC connections for the analog signals and the synchronization signal. It is possible to connect up to four USB A/D boards in a daisy chain.

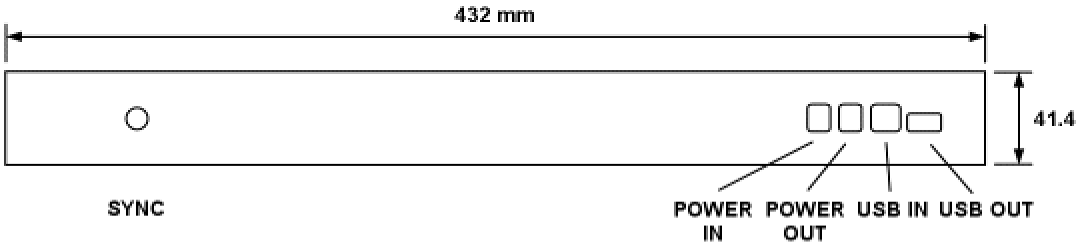


The following connections are available on the front view of the board:

CH. 1 - CH. 16
BNC connections for the 16 analog channels.

DIGITAL I/O
Port for controlling a Kistler force plate, the cable can be ordered from Qualisys.
Note: The port can also be used to control other applications. Pin 1-8 is then the digital I/O and pin 9-15 is ground.

EXTERNAL TRIGGER
BNC connection for the synchronization cable, see chapter "Hardware synchronization" on page E - 13.



The following connections are available on the rear view of the board:

SYNC
BNC connection for synchronization of several USB A/D boards.

POWER IN
Connection for the power supply, which is supplied by Qualisys.

POWER OUT
Connection for distributing the power to another USB A/D board.

USB IN
USB connection to the measurement computer. The cable is supplied by Qualisys.

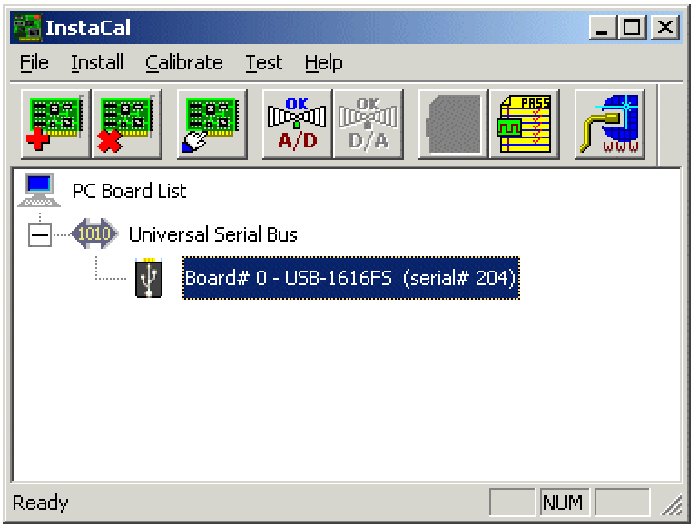
USB OUT
USB connection for connecting to another USB A/D board.

Installing the USB-1616FS board

Follow this procedure to install the USB A/D board:

1. Connect the supplied power supply to the **POWER IN** connection
2. Connect the **USB IN** port to an USB port on the measurement computer.
3. Wait 30 seconds while the computer detects the new hardware and installs it. If you look in the Device manager it will be installed as a Human Interface Device.
4. Run the Instacal software on the **Windows Start** menu.
5. Instacal will detect the USB A/D board and display the following dialog.

6. Click **OK**. The board will then be listed in Instacal as shown below.



7. Exit Instacal, you do not need to do any configurations for the USB A/D board.

When the board is properly installed it will be listed on the **Analog boards** page in the **Project options** dialog in QTM.

Note: If you have several USB A/D boards you must follow the instructions in chapter “Installing several USB-1616FS boards” below.

Installing several USB-1616FS boards

When using several USB boards they are connected in a daisy chain. The internal synchronization of the boards is done via the **SYNC** port on the rear view of the board. The first board must then be set to send the synchronization signal. Because the synchronization of the two boards is done internally, the synchronization cable from the master camera is only connected to the **EXTERNAL TRIGGER** port of the first USB A/D board.

When you use several USB A/D boards it is important that the board which is connected to the computer is the same as the first board of the list in Instacal. This is because the first board is used to send the synchronization signal. Therefore you should install the boards according to the following procedure:

1. Connect the boards in the daisy chain. If the boards comes in a rack, they are already daisy-chained so you ca go directly to step 2.

    a. Connect the power to the second board, by connecting the supplied power cord between the **POWER OUT** connection on the first board to the **POWER IN** connection on second board. Or connect a second power supply to the second board.

    b. Connect the **USB IN** port of the second board to the **USB OUT** port of the first board.

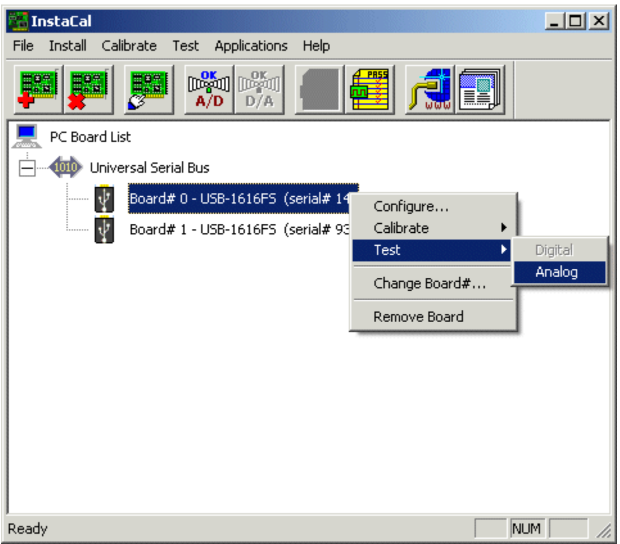
    c. Connect the rest of the daisy chain in the same way.

2. Connect the **USB IN** port of the first board to an USB port on the measurement computer.

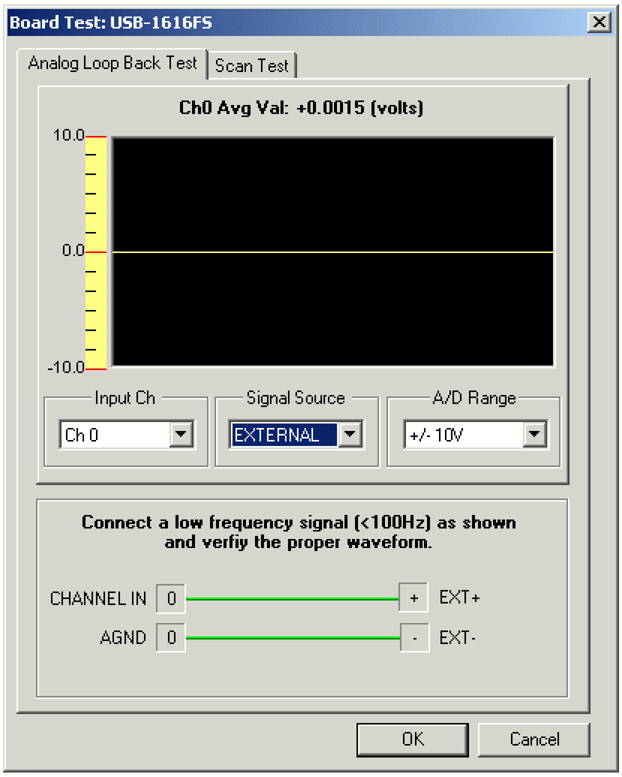
3. Follow the steps 3 to 6 in the chapter “Installing the USB-1616FS board” on page E - 9.

4. Finally make sure that the board numbers are correct. It is easiest to use the boards if the order in Instacal matches the order of the daisy chain. First you must set the first board to board number 0 in Instacal, so that the trigger signal from the cameras are connected to the correct camera.

    a. Right-click on the first USB board and click on **Test** and then **Analog**.



    b. Instacal starts to collect data from the board. Set the **Signal Source** to **EXTERNAL**. Then connect something to the first channel on the first board in the chain, **Ch 0** is the same as Ch. 1 on the front of the USB board.



If the signal changes, it is the correct board that is **Board# 0** and you can go on with step 5. If the signal does not change close the **Board test** window and then make an analog test of the other boards until you find the board that is connected first in the chain. When you have found the first board in the chain, follow these steps.

i. Right-click on the **Board# 0** in Instacal. Click on **Change Board#**.

ii. Click **OK** to change the board number.

iii. Right-click on the board that is the first board in the chain. Click on **Change Board#**.

iv. Click **OK** to change the board number to 0.

v. Finally go back to the board that was **Board 0#** and change the board number to the first available number.

5. Go on and test the rest of the boards if you have more than two boards. The rest of the boards do not have to be in the correct order, but it is easier to identify the channels if the boards are in the same order in Instacal as in the daisy chain. Change the board numbers with the procedure above so that the boards are in the correct order.

6. Connect the **SYNC** ports of the boards with a coaxial cable.

7. Connect the synchronization cable from the master camera to the **EXTERNAL TRIGGER** port of the first USB A/D board. The boards can now be used in QTM.

Important: If the boards are disconnected from each other, the same board as before must be the first board when they are connected to the computer. If you do not know which board that was first before, it is best to disconnect the boards from the computer and run Instacal. They will then be uninstalled and you can follow the instructions above to install the boards again.

Performance of USB-1616FS

A single USB-1616FS has the following specified maximum sample frequencies.

Number of Input Channels	Per-channel Throughput (Hz)
1	50000
2	50000
3	36000
4	30000
5	25000
6	22000
7	19000
8	17000
9	15000
10	14000
11	12500
12	12000
13	11250
14	10500
15	10000
16	9500

However the maximum frequency is lowered if the computer is running other programs at the same time. Therefore tests have been made to show the performance of the USB-1616FS board on different computers. The test also covers up to four boards connected to the same computer.

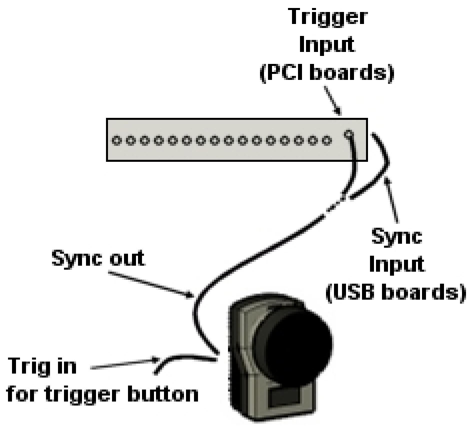
IMPORTANT DISCLAIMER! The numbers below are only examples of what can be achieved the actual measurement frequency on your computer and measurement setup must be measured.

	Windows 2000 desktop	**Windows XP desktop**	**Windows XP Laptop**
With preview frames			
16 channels	1680	4080	2520
32 channels	1560	3480	2280
64 channels	1440	2880	2040
With preview frames and webcamera			
64 channels	960	2400	2040

Hardware synchronization

In order to establish full synchronization between the camera system and the A/D board, hardware synchronization is used. Therefore a synchronization cable is connected between the camera system and the analog board. A synchronization cable which can be used to connect the analog board is supplied by Qualisys AB.

Note: If more than one analog board is installed in the computer, the analog synchronization cable must be connected to all of the analog board. Except for the USB-1616FS boards where it is only connected to the first analog board.



USB-2533 and USB-1616FS

For Oqus the recommended synchronization option is the frame synchronization, see chapter “External sync” on page 164. This option can also be used with ProReflex but then the analog frequency is always the same as the camera frequency. For this option you must connect the **Sync out** connection from the camera to the **Sync** connection on the back of the box.

The other option is to start the analog capture with the sync output from the camera. Then the frequency is set internally by the analog board, which means that there may be a small drift between the systems. This option must be used if you want a higher frequency on the ProReflex than the camera frequency. To use this option connect the **Sync out** connection from the camera to the **External trigger** connection on the front of the box

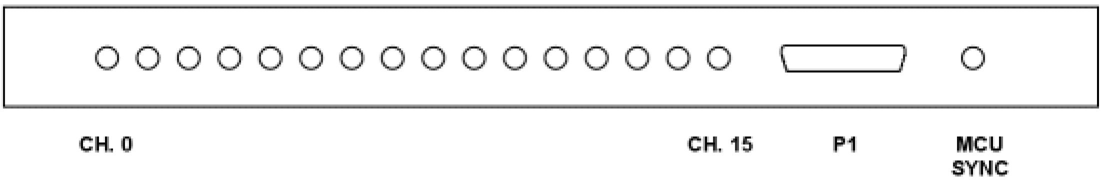
PCI-DAS6402/16 and PCI-DAS1602/16

For the PCI boards the only synchronization option is to start the analog capture with the sync output from the camera. Therefore the synchronization cable is connected between the **Sync out** connection of the camera system and the **External trigger** connection on the analog board. This connection is the BNC port furthest to the right on the front of the box.

Note: On a ProReflex system the control port cable must always be connected to the Master camera. For Oqus it is recommended to connect the cable to the Master camera if the synchronization signal is faster than 5000 Hz.

Note: The USB analog boards comes installed in a box with BNC connections. For the PCI analog boards it is recommended to use a BNC connection box for easier to connection of equipment to board, see chapter “BNC connection box” on the facing page. If you don't have a BNC connection box please contact Qualisys AB for details on the connection.

BNC connection box



The BNC connection box makes the connection to the PCI analog board much easier. The box has the following connections:

CH. 0 - CH. 16
BNC connections where the analog signals can be connected with BNC cables, instead of connecting the signals directly to the analog board.

P1
Not in use. Except if you are using pretrigger and external trigger with the PCI-DAS6402/16 board, then a jumper should be connected between pin 11 and pin 13.

MCU SYNC
BNC connection for the hardware synchronization signal.

Note: The cable between the BNC connection box and the analog board is supplied by Qualisys AB.

Appendix F: Wireless EMG systems

EMG systems in QTM

Any EMG (Electro Myo Graphy) system with analog output can be connected to an analog board and then captured in QTM. However the Noraxon and Mega EMG systems are integrated in QTM so that they can be collected directly without an extra analog board. The chapters “Noraxon EMG” below and “Mega EMG” on page F - 10 describes the settings and configurations that must be done to use these systems.

Noraxon EMG

QTM supports integration with the Noraxon TeleMyo 2400T G2 EMG and TeleMyo DTS. The TeleMyo 2400T G2 has wireless communication and 4 to 16 channels. A wide range of sensors can be connected to the channels. On the Noraxon TeleMyo DTS system each EMG channel consists of a wireless device.

The following instructions for Noraxon EMG will only cover how to connect to and capture data in QTM. For more detailed information on EMG and the TeleMyo 2400T G2 and Noraxon DTS please refer to the Noraxon manual.

Noraxon installation

Follow these instructions to install a Noraxon EMG on a new computer.

1. The files needed for the Noraxon is included in the QTM installer. If you cannot see the **Noraxon EMG** page in the **Project options** dialog, run the installer again and make sure to include the Noraxon files.
2. Connect the TeleMyo 2400 G2 PC Interface, TeleMyo 2400R G2 Mini Receiver or the TeleMyo DTS according to the description in chapters “Noraxon computer connection” on next page and “Noraxon trigger connection” on next page.
3. Start the Noraxon EMG and wait for the communication to be up and running.
4. Go to the **Noraxon EMG** page in the **Project options** dialog. Select **Use Noraxon EMG** and then click on **Device setup**.
5. Select the **Driver** in the dialog and click OK. The driver is always called **G2 USB/WLAN**.

a. If you want to check the connection click on **Setup** and then **Test Transmitters** to see a status report.

Noraxon computer connection

Four different interfaces can be used for the wireless communication with the Noraxon EMGs.

TeleMyo 2400 G2 PC Interface
Used with TeleMyo 2400T G2 EMG.

TeleMyo 2400R G2 Mini Receiver and **TeleMyo 2400R G2 Analog Output Receiver**
Can be used with TeleMyo 2400T G2 EMG and TeleMyo DTS.

Directly to the TeleMyo DTS
Recommended for TeleMyo DTS.

They are all connected via USB and use the driver called nxnusb.dll, which can be found in the Qualisys Track Manager folder. When connected the first time Windows will ask for the driver, make sure that you browse to the correct driver.

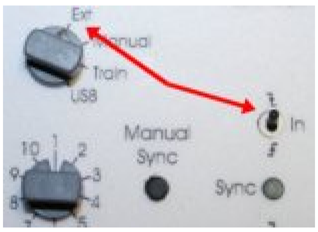
Noraxon trigger connection

Then to synchronize the Noraxon EMG system to the camera system a trigger signal must be sent to the system. How to connect the trigger differs slightly between the three interfaces, see instructions in the chapters below.

Trigger connection to TeleMyo 2400 G2 PC Interface

Follow these instructions to connect the trigger used for the TeleMyo 2400 G2 PC Interface. To send the trigger signal to the TeleMyo 2400T G2 EMG you need a Wireless Transmitter and Sync Receiver.

1. Do NOT turn on the Wireless Transmitter before the last step.
2. Connect the Wireless Sync Receiver to the **Sync in** port on the Noraxon system.
3. Connect the Wireless Transmitter to the **External trigger input** connector on the **Control** port cable of the camera system. Use the wire with a phono connection in one end and a BNC connection in other end to connect from **In** to the **External trigger input**.
   a. Set the settings on the Transmitter to **Ext** and negative edge on the **In** connection.

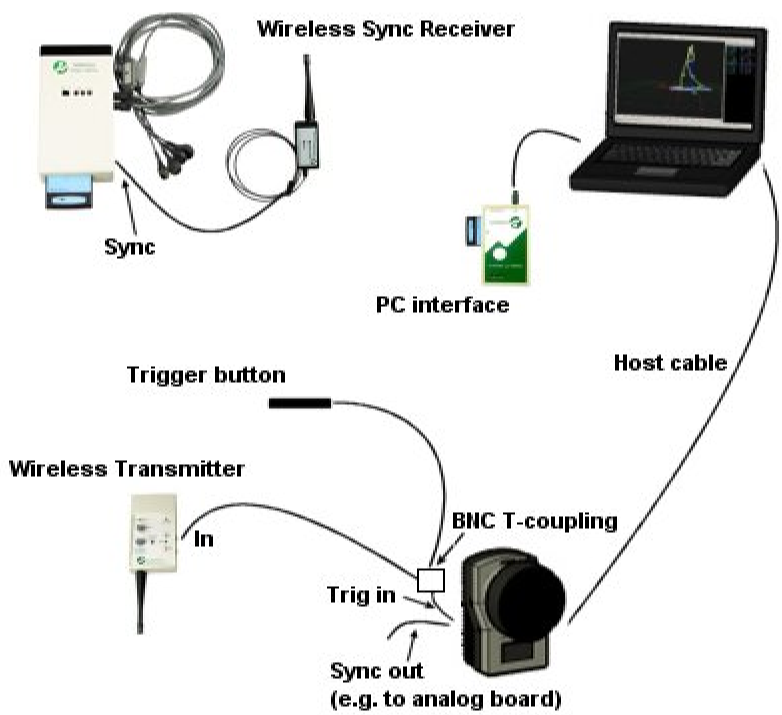


   b. Put a BNC T-coupling on the **External trigger input** so that you can also connect a trigger button to the input. The measurement must be started with the trigger button.
   c. The camera must be set to trigger on negative edge, which is done on the **Advanced Timing** page in the **Project options** dialog This can only

be changed for Oqus cameras. ProReflex cameras always triggers on negative edge.

d. On a ProReflex camera the **Control** port cable must be connected to the Master camera, i.e. the first camera in the daisy chain.

4. Make sure that the Noraxon EMG device is turned on and connected before starting the Wireless Transmitter. Test the batteries on the Transmitter by selecting **Manual** and clicking on the **Manual Sync** button. Then the **Sync** LED on the Transmitter and on the Wireless sync receiver should be lit. Remember to switch back to **Ext** after testing.
Note: If both LEDs are lit on the Wireless trigger before pressing the trigger button, then you have selected the wrong polarity on the Wireless Transmitter. If the **Signal quality** LED is orange then there is no wireless connection to the Wireless Transmitter, check that it is turned on.

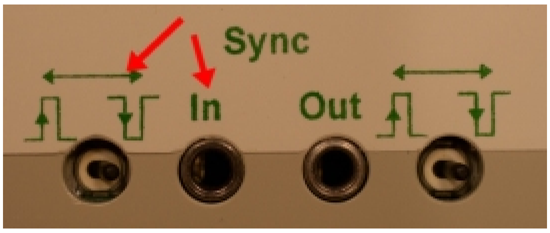


Trigger connection to TeleMyo 2400R G2 Mini Receiver or TeleMyo 2400R G2 Analog Output Receiver

Follow these instructions to connect the trigger used for the TeleMyo 2400R G2 Mini Receiver or TeleMyo 2400R G2 Analog Output Receiver. You also need Wireless Sync Receiver to receive the trigger signal on the TeleMyo 2400T G2 EMG.

1. Connect the Wireless Sync Receiver to the **Sync in** port on the Noraxon system.
2. Connect the Receiver interface to the **External trigger input** connector on the **Control** port cable of the camera system. Use the wire with a phono connector in one end and a BNC connector in other end to connect from **In** to the **External trigger input**.

   a. Set the settings on the Receiver interface to negative edge for the **In** connection.

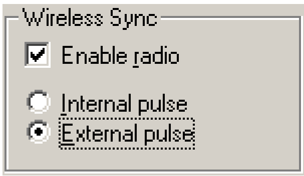


b. Put a BNC T-coupling on the **External trigger input** so that you can also connect a trigger button to the input. The measurement must be started with the trigger button.

c. The camera must be set to trigger on negative edge, which is done on the **Advanced Timing** page in the **Project options** dialog This can only be changed for Oqus cameras. ProReflex cameras always triggers on negative edge.

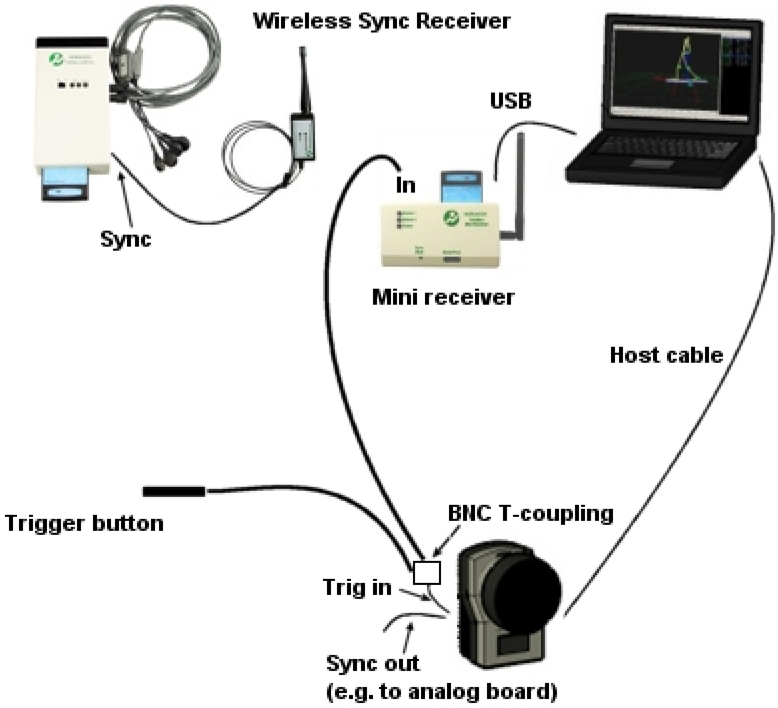
d. On a ProReflex camera the **Control** port cable must be connected to the Master camera, i.e. the first camera in the daisy chain.

3. To activate the trigger input on the Receiver interface you must go to the **USB receiver configuration dialog** in QTM, see chapter “Control panel” on page F - 9. Select the **Enable radio** and **External pulse** settings.



4. Start the Noraxon EMG device and then test trigger connection by pressing the trigger button. Then the **Sync** LED on the front of the Receiver interface and on the Wireless sync receiver should be lit. You do not need to start a measurement in QTM but the cameras must be turned on.

Note: If both LEDs are lit on the Wireless trigger before pressing the trigger button, then you have selected the wrong polarity on the Receiver interface. If the **Signal quality** LED is orange then there is no wireless connection to the Receiver interface, check that the radio is enabled, see point 3 above.



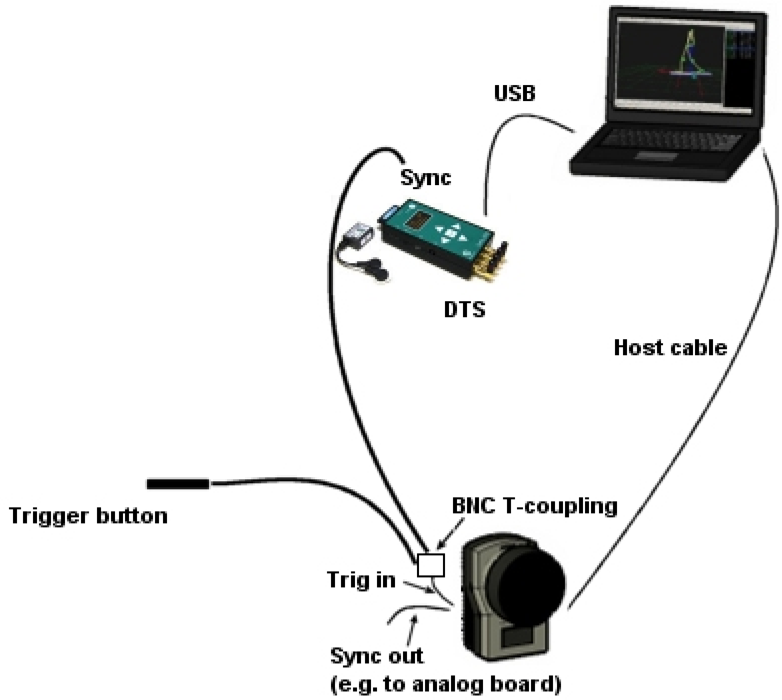
Trigger connection to TeleMyo DTS

The recommended trigger connection for the TeleMyo DTS system is to connect the trigger signal directly to the DTS unit. Even if it works with the Mini receiver as well.

1. Split the trigger button signal with a T-coupling and then connect the signal to **Sync** input on the DTS system. The cable is different from the one used for the other Wireless transmitter and Mini receiver. There is a small black box on the middle of the wire.

2. For the DTS system you will also need to assign the sensors, for instructions see the Noraxon manual.

3. Test the system by making a measurement in QTM, see chapter "Making a measurement with Noraxon".

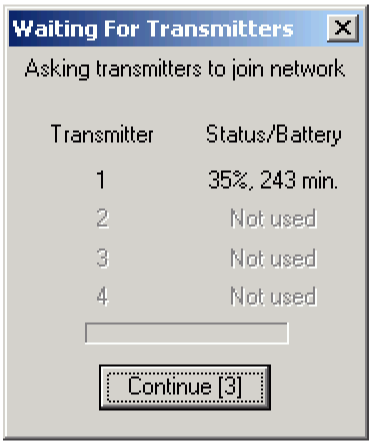


Making a measurement with Noraxon

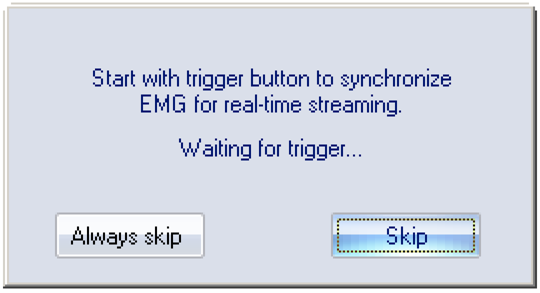
With the Noraxon EMG it is possible to capture up to 16 EMG channels or other type of sensor data, depending on the model. Follow these steps to make a measurement with Noraxon EMG in QTM.

1. The first time a Noraxon EMG is connected to a computer you must follow the instructions in the chapter “Noraxon installation” on page F - 1.
2. Before starting a new measurement connect the synchronization devices according to the description in chapter “Noraxon trigger connection” on page F - 2.
3. Open the **Project options** dialog and go to the **Timing** page. Set the camera system to start on external trigger.
4. Go to the **Noraxon EMG** page and select the settings below, for more information see chapter “Noraxon QTM settings” on page F - 8.
5. Select the **Use Noraxon EMG** option to activate the EMG capture. Remember to uncheck this option when the Noraxon EMG is turned off.
   Note: Select the **TeleMyo DTS** option if you are using a DTS system.
6. Select the number of **Channels** on the Noraxon EMG. This is the total number of channel on the EMG. Do not select the wrong number of channels as it can result in problems with the synchronization. The number of channels used in a measurement is selected in the **Analog channels** list.
7. Activate the channels in the list to the right by selecting the channels. Do not activate channels that are not connected as that will only result in larger files.
8. Close the **Project options** dialog and start a new measurement. The dialog below will be displayed for a few seconds while the measurement is prepared

on the Noraxon EMG. This dialog will also appear when changing settings while in preview and when opening the **Start capture** dialog make sure that you never click **Cancel** in the dialog.



9. When **Use trig in Real Time** the Noraxon data must be triggered by an external trigger signal to be synchronized in real time. Then the dialog below will appear every time you start a new measurement and every time you change something in the **Project options** dialog. If you do not use the real-time output from QTM disable the **Use trig in Real Time** on the **Noraxon EMG** page.



10. Open the **Data info window** on the **View** menu and then right-click and select **Display Analog data**.

11. The data from the Noraxon EMG is listed with the analog data. The channels are called **EMG Chan 1** and so on. The **Board name** is Noraxon.

12. When starting a capture the start of the EMG data will be synchronized with the motion data by the external trigger signal.

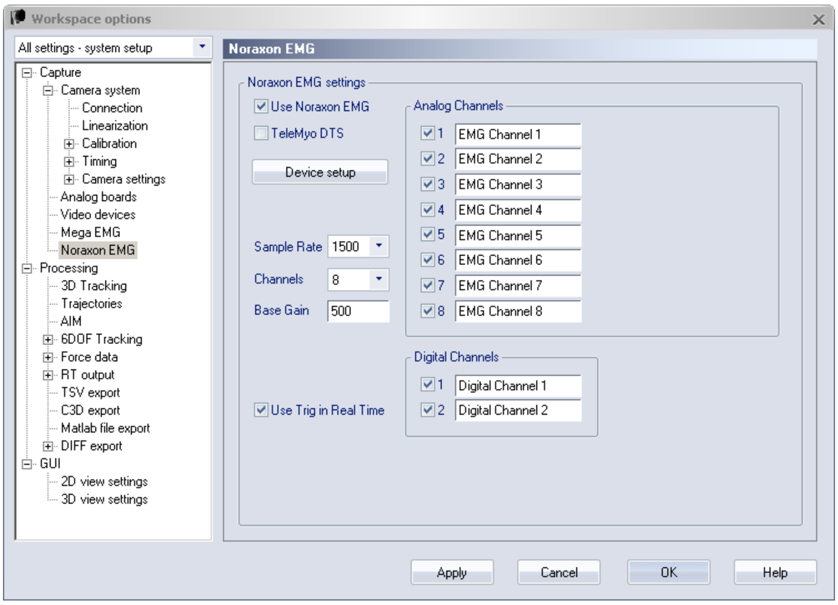
Export Noraxon EMG data

When the EMG data is exported to another format (C3D, TSV or to Matlab) the data is resampled so that the frequency is a multiple of the camera capture rate. The new sampling rate will therefore be as close as possible to the EMG frequency, e.g. if the camera capture rate is 120 Hz and the EMG is 1500 Hz the new sample rate will be

1440 Hz. If the analog capture rate is higher than the EMG rate the new sample rate will be the same as the analog capture rate.

Noraxon QTM settings

The **Noraxon EMG** page is included in the **Project options** dialog when the additional files for Noraxon have been installed using the QTM installer.



The following settings are available on the page:

Use Noraxon EMG
To activate the Noraxon EMG check the **Use Noraxon EMG** option. It is important to uncheck this option if the Noraxon EMG is not connected otherwise QTM will try to connect but fail, which will result in errors.

TeleMyo DTS
Activate the **TeleMyo DTS** option if you have a DTS system, so that the system will work properly.

Device setup
Click on **Device setup** to select the device and change the settings, see chapter "Noraxon driver setup" on the facing page.

Analog channels
Select the number of channels on the Noraxon EMG under the **Active channels** heading activate the channels that are in use on the EMG. You can enter a name for the signal next to the signal, the name will be included in the export. Do not activate a channel that you do not use as it will only result in a larger file.

Digital channels
You can also use the **Digital channels** to collect data for example from a foot switch. There are 2 digital channels on both the TeleMyo and the DTS system. For more information check the Noraxon manual.

Sample rate
The **Sample rate** is specified in Hz. **1500** is sufficient for most EMG measurements.

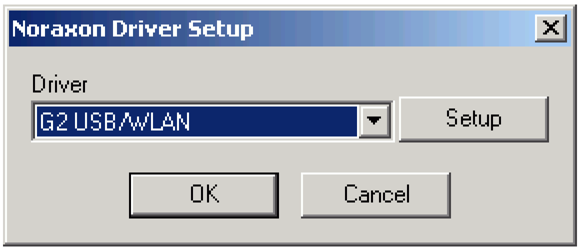
Base gain
Set the **Base gain** of the EMG sensors, the standard base gain is 500 times.

Use Trig in Real Time

The **Use trig in Real Time** option activates the use of the trigger button in Real-time mode. When the option is activated the real-time will not start until you press the trigger button. This is to ensure the synchronization in real-time. If you do not use the real-time output you can uncheck the option and then you only need to press the trigger button before the measurement.

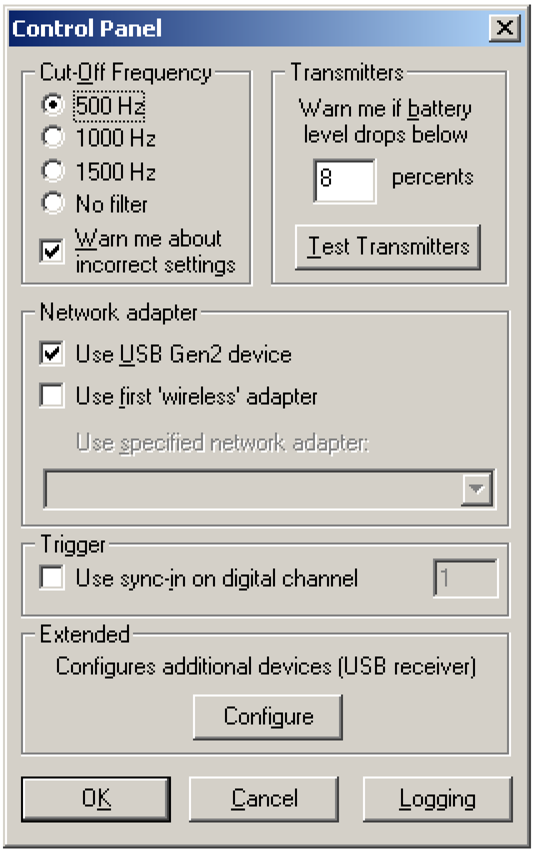
Note: Activate the **Use trig in Real Time** option when the setup is finished to minimize the number of times you have to press the trigger button.

Noraxon driver setup



The **Noraxon driver setup** dialog is used to select the driver for the communication. For information on the options see the Noraxon manual. For all of the interfaces supported by QTM the driver must be **G2 USB/WLAN**. Click **Setup** to for more options, see chapter “Control panel” below.

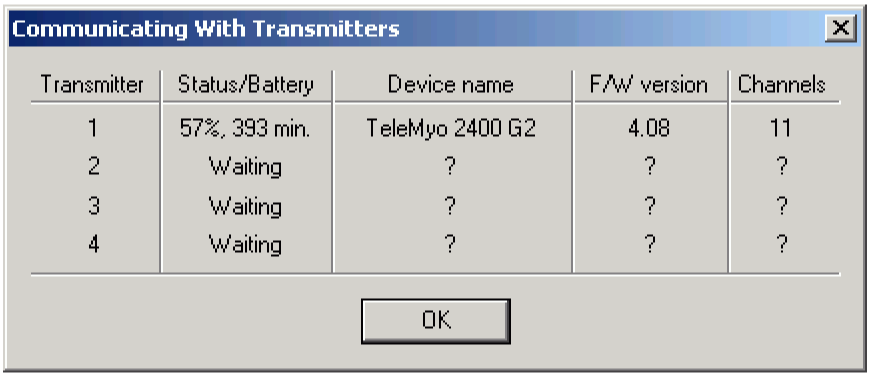
Control panel



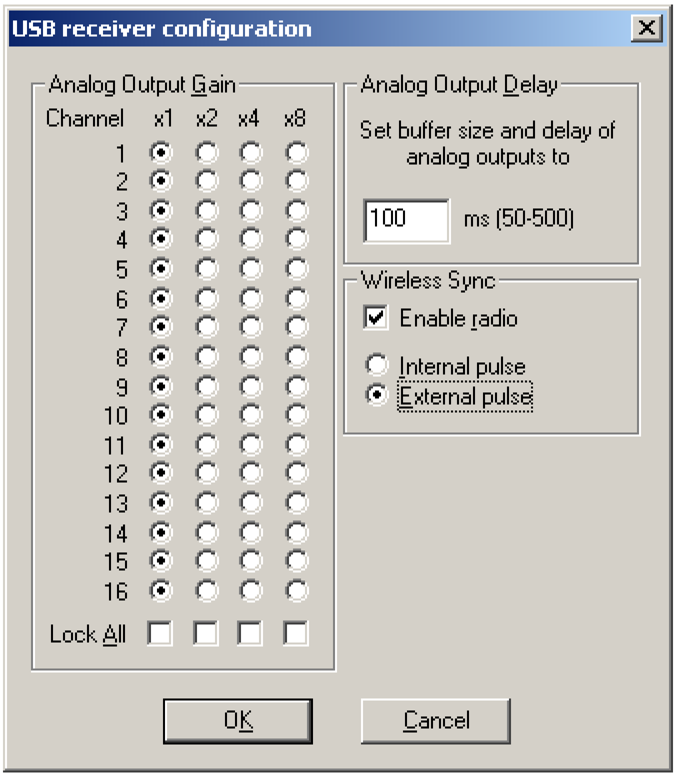
The settings on the **Control panel** are imported from the Noraxon program, see the Noraxon manual for more information.

Make sure that the **Network adapter** setting is set to **Use USB Gen2 device**. If you have to change it from **Use first 'wireless' adapter** you must close and open the dialog before you can test the transmitter. The other settings must be as displayed in the image above.

Click on **Test transmitters** to open the dialog below which displays the status of the connected Noraxon unit.



⚠ **IMPORTANT:** For the Mini Receiver you must click on **Configure** to open the **USB receiver configuration**. Then activate the sync with the **Enable radio** and **External pulse** settings.



Mega EMG

QTM supports integration with the Mega ME6000 telemetry and data logger. The ME6000 has wireless communication and 4 to 16 channels. A wide range of sensors can be connected to the channels.

The following instructions for Mega EMG will only cover how to connect to and capture data in QTM. For more detailed information on EMG and the ME6000 please refer to the Mega manual.

The Mega integration does not work on Windows 7, because we have no fully functional driver from Mega.

Mega installation

Follow these instructions to install a Mega EMG on a new computer.

1. The files needed for the Mega EMG is included in the QTM installer. If you cannot see the **Mega EMG** page in the **Project options** dialog, run the installer again and make sure to include the Mega driver.

   Note: The drivers cannot be installed on Windows 7, since there is no fully functional driver supplied by Mega.

2. Configure the wireless connection according to the instructions in chapter “Mega WLAN configuration” below.

3. Connect the Mega EMG to QTM according to the instructions in chapter “Connecting to Mega in QTM” on page F - 13.

Mega WLAN configuration

A wireless connection must be configured before the Mega EMG system can be connected. To work with QTM it must be configured in the following way.

1. First install the drivers for the WLAN USB stick from the cd that is included in the Mega case.

2. Insert the WLAN USB stick and follow the installation instructions.

3. Start the Mega EMG and go to **Preferences**/**Wireless**/**IP Parameters**. Note the IP and the router addresses so that you can specify the computer IP address later.

   The Mega EMG should have a WLAN profile called MegaTest that has the configuration according to the list below, but it may have been changed. Check the profile on the ME6000 on **Preferences**/**Wireless**/**WLAN parameters**. If the profile is incorrect you must change a profile, read the Mega manual on how to change a profile.

   Profile name
   Select any name that you want.

   SSID
   Select any ID that you want.

   Wireless mode
   Ad-hoc

   Transmit rate
   Auto

   Channel
   Select any channel.

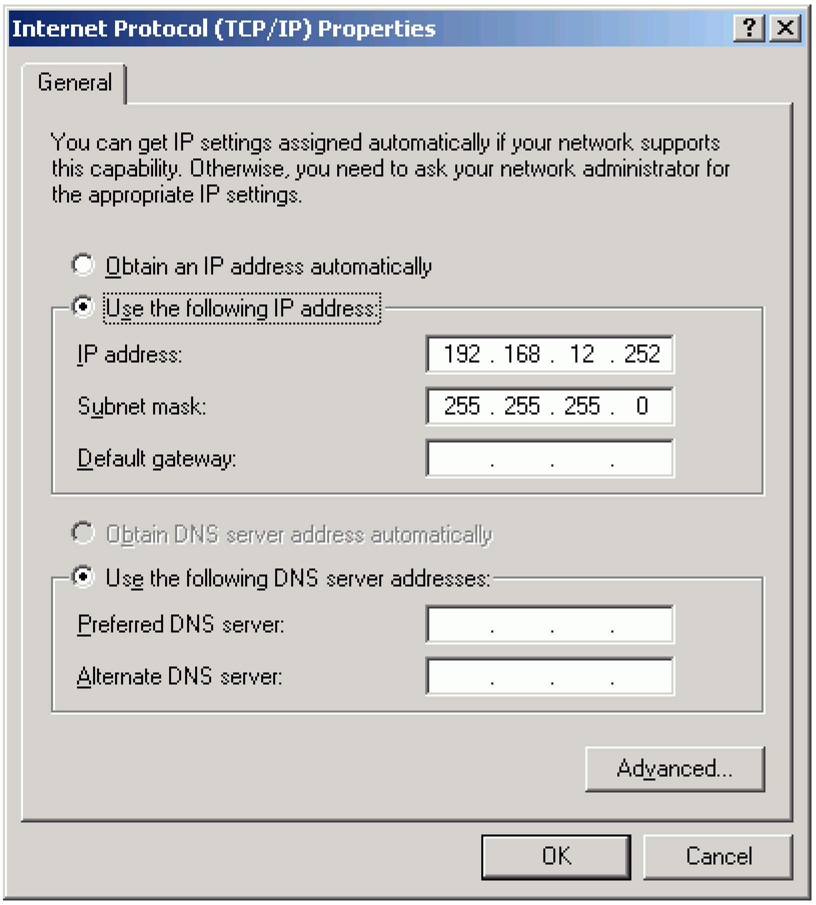
   Data encryption
   Not activated

   ME6000 IP address
   The IP number can be selected freely, so keep the address if the profile already got one. Otherwise use for example 192.168.12.253. If that net already exist change the third number from 12 to another number.

   Subnet mask
   255.255.255.0

PC IP / Default gateway
The three first number must be the same as ME6000 IP address the last number must be 1.

4. Open the **Network connections** window from **Settings** on the **Windows Start** menu.
5. Select the Wireless network that is associated with the USB stick and rename it to Mega.
6. Right-click on the Mega network and open the **Properties**. Scroll down the list and double-click on **Internet Protocol (TCP/IP)**.



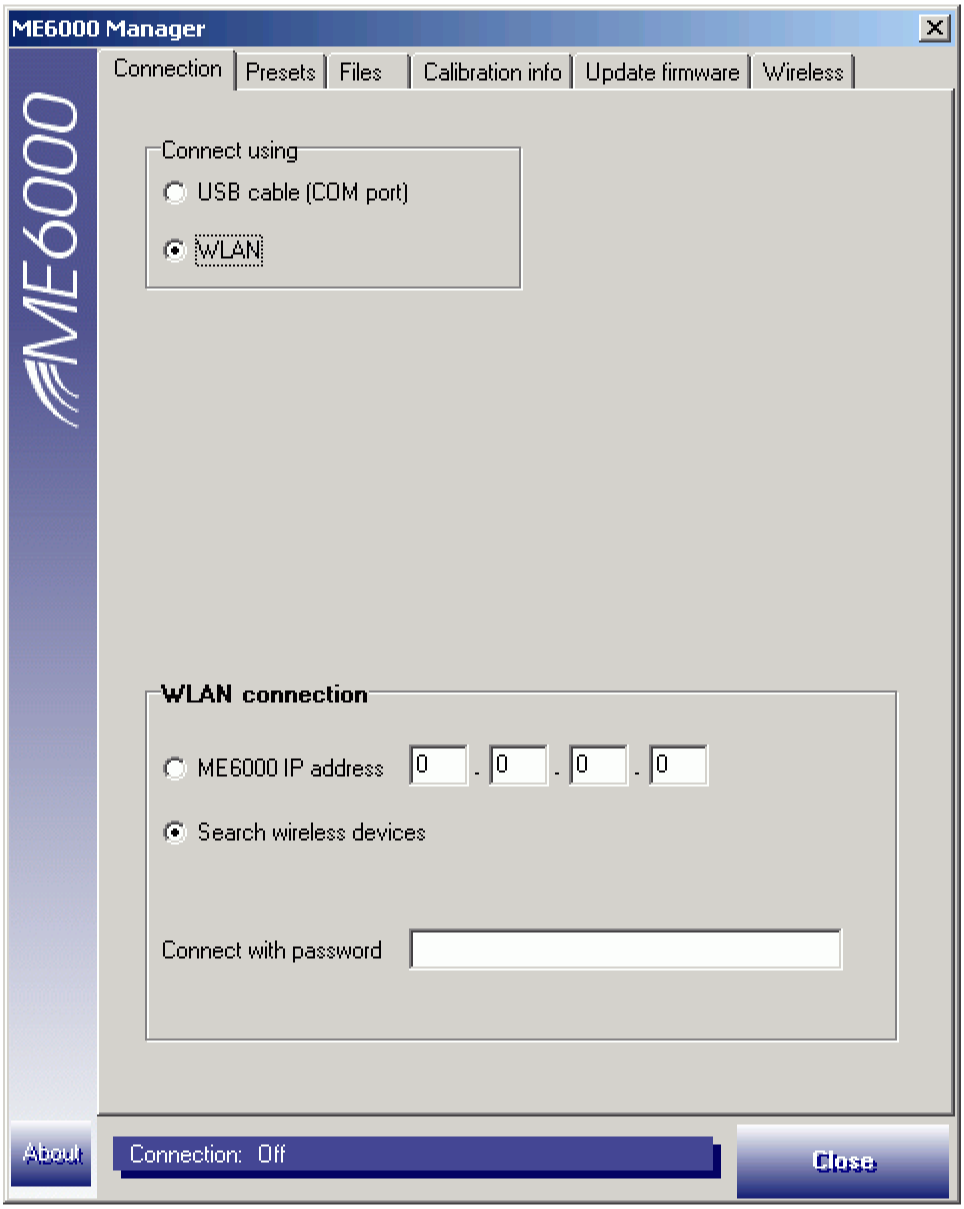
7. Change the IP setting to **Use the following IP address**.
   a. Set the same net as in **IP Parameters** on the Mega EMG. That means that the first three numbers must be the same and that the last number must be different from the Mega EMG and its router.
   b. Set the **Subnet mask** to 255.255.255.0.
   c. Click **OK** twice.
8. Now you can connect to the Mega EMG. Open the program that manages the wireless network. The easiest way is to right-click on the wireless network symbol in the Windows status bar. If the Mega network is managed by another program you must use that to connect to the network.
   a. If you cannot see the network click **Refresh network list**.
   b. When the network is visible, select it and click **Connect**. Wait until it is connected.

9. The next step is to connect to the Mega EMG in QTM, see chapter "Connecting to Mega in QTM" below.

Connecting to Mega in QTM

When the WLAN has been configured the next step is to connect to the Mega EMG in QTM.

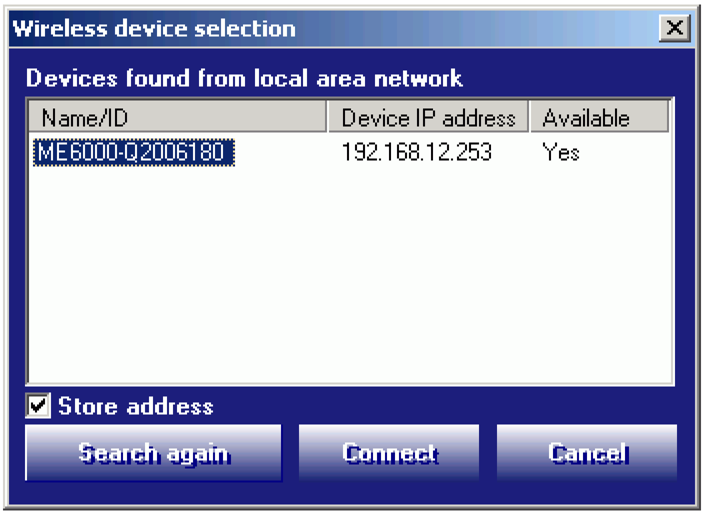
1. Start QTM and open the **Project options** dialog.
2. Go to the **Mega EMG** page.
3. Click on **ME6000 Manager** to open the dialog with Mega's internal settings. For detailed descriptions please refer to the Mega manual. These instructions will only describe how to connect to the Mega in QTM.
4. Select **WLAN** and **Search wireless device** on the **Connection** tab in the **ME6000 Manager** dialog.



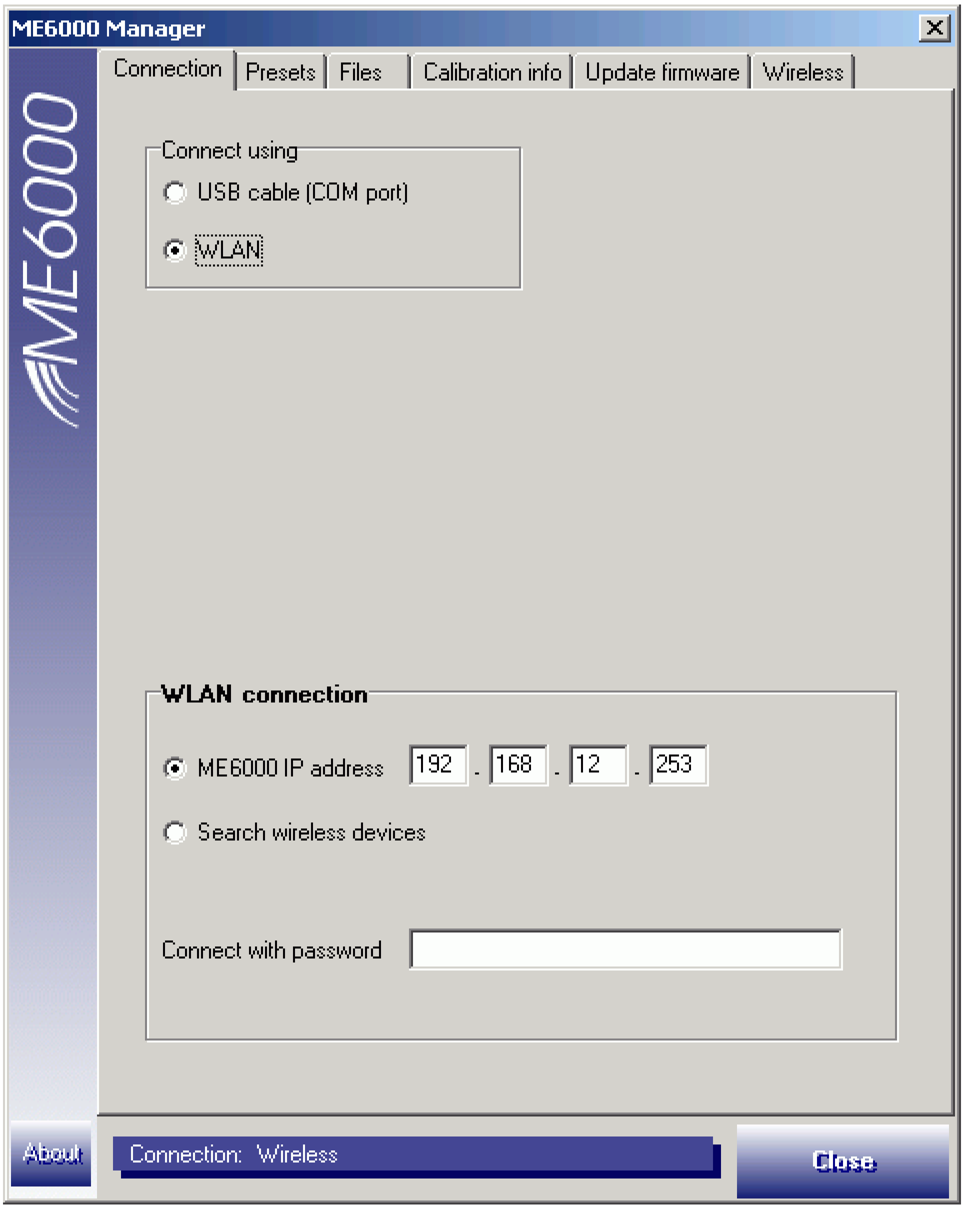
5. Click on any other tab to open the **Wireless device selection** dialog. If the EMG is not found click on **Search again**.

   a. When the device is found, select it so that the text turns blue.

b. Then select the option **Store address** so that the IP address is saved to the next session. Otherwise you must perform the connection process every time you shutdown the device.



6. Click **Connect** and then go to the **Connection** tab again, the IP address of the EMG will be shown in **ME6000 IP address**. The connection is finished and you can close the **ME6000 Manager** dialog.



Mega hardware connections

To synchronize the Mega EMG system to the camera system the Video RF Trigger and Receiver boxes from Mega must be connected to the camera system respectively to the Mega system. Follow these instructions:

1. Connect the Video RF Receiver to the **Tr/in** on the Mega system. Use the short wire to connect from **Pulse** to **Tr/in**.

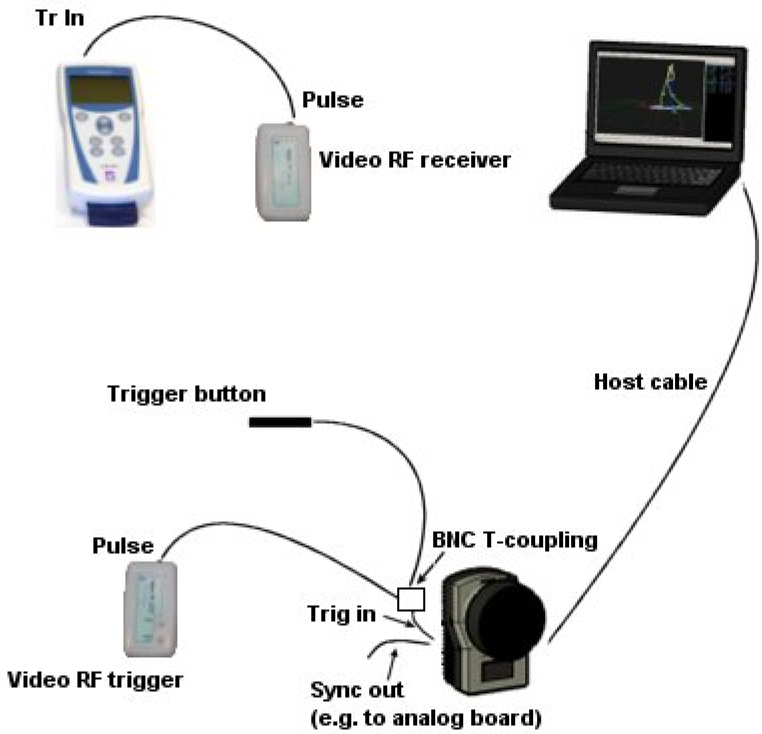
2. Connect the Video RF Trigger to the **External trigger input** connector on the **Control** port cable of the camera system. Use the long wire with a BNC connection in one end to connect from **Pulse** to the **External trigger input**.

    a. Put a BNC T-coupling on the **External trigger input** so that you can also connect a trigger button to the input. The measurement must be started with the trigger button.

    b. The camera must be set to trigger on negative edge. This can only be changed for Oqus cameras. ProReflex cameras always triggers on negative edge.

c. On a ProReflex camera the **Control** port cable must be connected to the Master camera, i.e. the first camera in the daisy chain.

3. Turn on the RF devices with the switch next to the **Pulse** connector. Test the batteries by clicking on the **Start** button on the RF Trigger and check that both yellow LEDs on the devices are blinking.

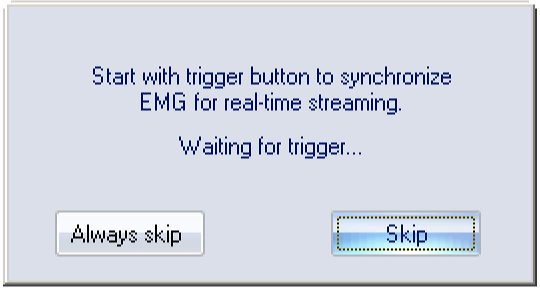


Making a measurement with Mega

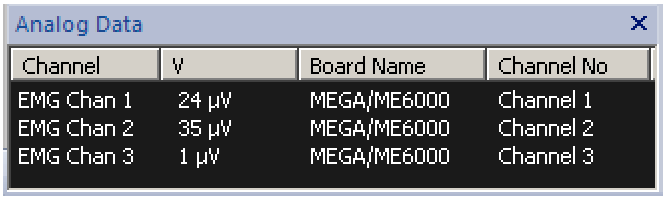
With the Mega ME6000 it is possible to capture up to 16 EMG channels or other type of sensor data, depending on the ME6000 model. Follow these steps to make a measurement with Mega EMG in QTM.

1. The first time a ME6000 is connected to a computer you must follow the instructions in the chapter “Mega installation” on page F - 11.
2. Before starting a new measurement connect the Video RF devices according to the description in chapter “Mega hardware connections” on previous page.
   a. Test the batteries by clicking on the **Start** button on the RF Trigger and check that both yellow LEDs on the devices are blinking.
3. Open the **Project options** dialog and go to the **Timing** page. Set the camera system to start on external trigger.
4. Go to the **Mega EMG** page and select the settings below, for more information see chapter “Mega QTM settings” on page F - 18.
5. Select the **Use Mega EMG** option to activate the EMG capture. Remember to uncheck this option when the Mega EMG is turned off.
6. Select the model of Mega EMG. The model depends on the number of channels on the EMG.

7. Activate the channels in the list to the right by selecting the type of sensor that is connected. Do not activate channels that are not connected as that will only result in larger files.
8. The options **Measuring form** and **Sampling rate** can be kept on **Raw** respectively **1000/1020**.
9. Close the **Project options** dialog and start a new measurement. If there is no connection with the Mega EMG two error messages will appear, see chapter “Mega measurement unit not responding” below.
10. When **Use trig in Real Time** the MEGA data must be triggered by an external trigger signal to be synchronized in real time. Then the dialog below will appear every time you start a new measurement and every time you change something in the **Project options** dialog. If you do not use the real-time output from QTM disable the **Use trig in Real Time** on the **MEGA EMG** page.



11. Open the **Data info window** on the **View** menu and then right-click and select **Display Analog data**.
12. The data from the Mega EMG is listed with the analog data. The channels are called **EMG Chan 1** and so on. The **Board name** is MEGA/ME6000.

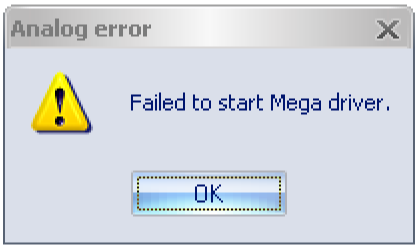


13. When starting a capture the start of the EMG data will be synchronized with the motion data by the external trigger signal.

Mega measurement unit not responding

When there is no communication with the Mega EMG during preview the following two error messages will appear. They will sometimes continue to appear even after the preview mode has been closed down, because they continue to be sent from the Mega driver.

QTM will try to restart the communication with Mega when these error occurs. If it does not start the following message will appear and the ME6000 will be inactivated on the **Mega EMG** page.



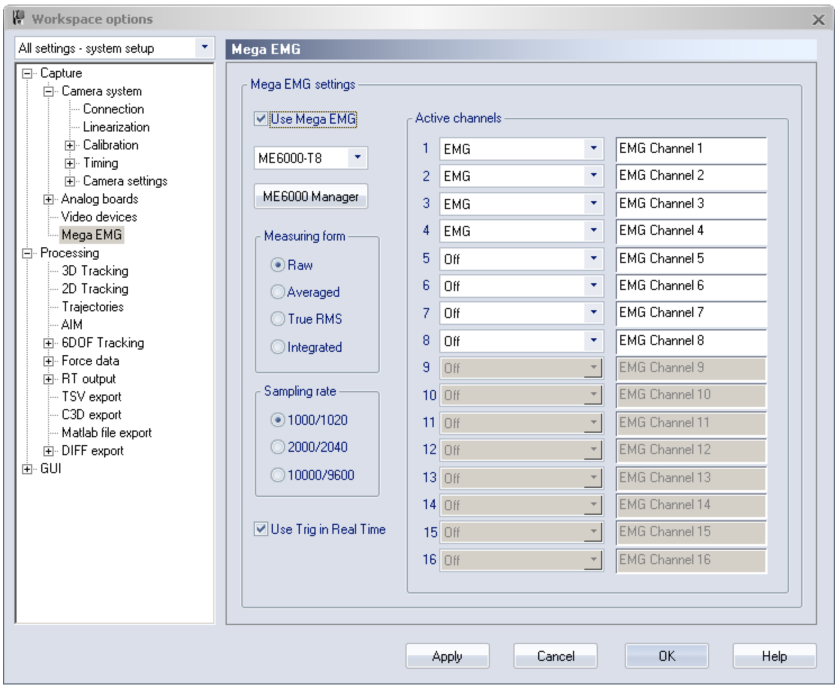
To fix the problem check that the WLAN configuration according to the instructions in chapter "Mega WLAN configuration" on page F - 11. If you do not want to use Mega in the capture, you must uncheck **Use Mega EMG** on the **Mega EMG** page in the **Project options** dialog.

Export Mega EMG data

When the EMG data is exported to another format (C3D, TSV or to Matlab) the data is resampled so that the frequency is a multiple of the camera capture rate. The new sampling rate will therefore be as close as possible to the EMG frequency, e.g. if the camera capture rate is 100 Hz and the EMG is 1020 Hz the new sample rate will be 1000 Hz. If the analog capture rate is higher than the EMG rate the new sample rate will be the same as the analog capture rate.

Mega QTM settings

The **Mega EMG** page is included in the **Project options** dialog when the additional files for Mega have been installed in the Qualisys Track Manager folder.



To activate the Mega EMG check the **Use Mega EMG** option. It is important to uncheck this option if the Mega EMG is not connected otherwise QTM will try to connect but fail, which will result in errors.

Select the Mega model, e.g. ME6000-T4, from the drop-down list. QTM cannot detect this so you must set it before starting a capture.

Click on **ME6000 Manager** to open **ME6000 Manager** dialog with more settings for the Mega system, see Mega manual for description of the settings.

Under the **Active channels** heading select the input on the EMG channels. You can enter a name for the signal next to the signal, the name will be included in the export. Do not activate a channel that you do not use as it will only result in a larger file and risk of slowing down the sampling rate. There are a lot of sensors except for EMG that can be connected to the Mega system, please contact Mega for more information.

The **Measuring from** list is used to select how the analog signal is measured. In most cases **Raw** is used because then the signal is unprocessed. All of the other settings will process the signal and change it from the original.

The **Sampling rate** is specified in Hz. **1000/1020** is sufficient for most EMG measurements. The two numbers is because Mega has two modes for the frequency, which can only be changed on the Mega EMG not from QTM. To check which mode you are using go to **Preferences**/**Sampling Freq** on the Mega EMG. The option **Standard** is dividable by 25 and corresponds to the first number, e.g. 1000. While the **Alternative** option is dividable by 30 and corresponds to the second number, e.g. 1020.

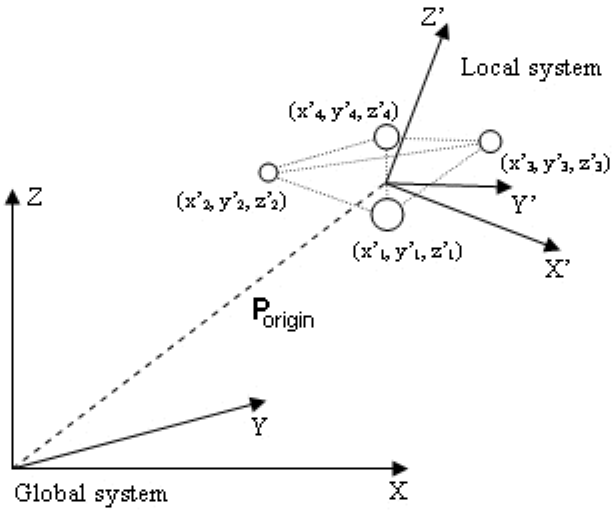
The sampling rate of the Mega EMG is limited by the number of channels. E.g. with 16 channels the maximum sampling rate is 1000 Hz. Higher sampling rates can be used with fewer channels: 8 channels at 2 kHz or 2-3 channels at 10 kHz.

The **Use trig in Real Time** option activates the use of the trigger button in Real-time mode. When the option is activated the real-time will not start until you press the trigger button. This is to ensure the synchronization in real-time. If you do not use the real-time output you can uncheck the option and then you only need to press the trigger button before the measurement.

Note: Activate the **Use trig in Real Time** option when the setup is finished to minimize the number of times you have to press the trigger button.

Appendix G: Rotation angle calculations in QTM

6DOF tracking output



The 6DOF tracking function uses the rigid body definition to compute $\mathbf{P}_{origin}$, the positional vector of the origin of the local coordinate system in the global coordinate system, and **R**, the rotation matrix which describes the rotation of the rigid body.

The rotation matrix (**R**) can then be used to transform a position $\mathbf{P}_{local}$ (e.g. x'_1, y'_1, z'_1) in the local coordinate system , which is translated and rotated, to a position $\mathbf{P}_{global}$ (e.g. x_1, y_1, z_1) in the global coordinate system. The following equation is used to transform a position:

$$\mathbf{P}_{global} = \mathbf{R} \cdot \mathbf{P}_{local} + \mathbf{P}_{origin}$$

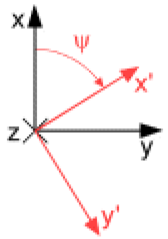
If the 6DOF data refer to another coordinate system than the global coordinate the position and rotation is calculated in reference to that coordinate system instead. The coordinate system for rigid body data is then referred to the global coordinate system.

Calculation of rotation angles from the rotation matrix (Qualisys standard)

The rotation angles are calculated from the rotation matrix (**R**), by expressing it in the three rotation angles: roll (θ), pitch (ϕ) and yaw (ψ).

To begin with the rotations are described with individual rotation matrixes: $\mathbf{R}_x$, $\mathbf{R}_y$ and $\mathbf{R}_z$. The rotations are around the X-, Y- respectively Z-axis and positive rotation is clockwise when looking in the direction of the axis.

The individual rotation matrix can be derived by drawing a figure, see example below. To get an individual rotation matrix express for example the new orientation x' and y' in the coordinates x, y and ψ and then make a matrix of the equations. The example below is of rotation around the Z-axis, where positive rotation of yaw (ψ) is clockwise when the Z-axis points inward.



The resulting three rotation matrixes are then:

$$\mathbf{R}_x = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\theta & -\sin\theta \\ 0 & \sin\theta & \cos\theta \end{bmatrix}$$

$$\mathbf{R}_y = \begin{bmatrix} \cos\phi & 0 & \sin\phi \\ 0 & 1 & 0 \\ -\sin\phi & 0 & \cos\phi \end{bmatrix}$$

$$\mathbf{R}_z = \begin{bmatrix} \cos\psi & -\sin\psi & 0 \\ \sin\psi & \cos\psi & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

The rotation matrix ($\mathbf{R}$) is then calculated by multiplying the three rotation matrixes. The orders of the multiplications below means that roll is applied first, then pitch and finally yaw.

$$\mathbf{R} = \mathbf{R}_x \cdot \mathbf{R}_y \cdot \mathbf{R}_z = \begin{bmatrix} r_{11} & r_{12} & r_{13} \\ r_{21} & r_{22} & r_{23} \\ r_{31} & r_{32} & r_{33} \end{bmatrix} =$$

$$= \begin{bmatrix} \cos\phi\cdot\cos\psi & -\cos\phi\cdot\sin\psi & \sin\phi \\ \cos\theta\cdot\sin\psi+\cos\psi\cdot\sin\theta\cdot\sin\phi & \cos\theta\cdot\cos\psi-\sin\theta\cdot\sin\phi\cdot\sin\psi & -\cos\phi\cdot\sin\theta \\ \sin\theta\cdot\sin\psi-\cos\theta\cdot\cos\psi\cdot\sin\phi & \cos\psi\cdot\sin\theta+\cos\theta\cdot\sin\phi\cdot\sin\psi & \cos\theta\cdot\cos\phi \end{bmatrix}$$

The following equations are then used to calculate the rotation angels from the rotation matrix:

$$\text{Pitch} : \phi = \arcsin(r_{13})$$

$$\text{Roll} : \theta = \arccos\left(\frac{r_{33}}{\cos\phi}\right)$$

$$\text{Yaw} : \psi = \arccos\left(\frac{r_{11}}{\cos\phi}\right)$$

The range of the pitch angle is -90° to 90°, because of the nature of the arcsin function. The range of the arcos function is 0° and 180°, but the range of roll and yaw can be expanded by looking respectively on the r_{23} and r_{12} elements in the rotation matrix (**R**). The roll and yaw will have the opposite sign compared to these elements, since cos(ϕ) is always positive when ϕ is within ±90°. This means that the range of roll and yaw are -180° to 180°.

A problem with the equations above is that when pitch is exactly ±90° then the other angles are undefined, because of the division by zero, i.e. singularity. The result of a pitch of exactly ±90° is that the Z-axis will be positioned at the position that the X-axis had before the rotation. Therefore yaw can be set to 0°, because every rotation of the Z-axis could have been made on the X-axis before pitching 90°. When yaw is 0° and pitch is ±90°, the rotation matrix can be simplified to:

$$\begin{matrix} 0 & 0 & \pm 1 \\ \pm\sin(\Theta) & \cos(\Theta) & 0 \\ \pm\cos(\Theta) & \sin(\Theta) & 0 \end{matrix}$$

From the matrix above the roll can be calculated in the range ±180°.

Important: With the definitions above, roll, pitch and yaw are unambiguous and can describe any orientations of the rigid body. However, when the pitch (ϕ) is close to ±90°, small changes in the orientation of the measured rigid body can result in large differences in the rotations because of the singularity at ϕ=±90°.

Calculation of other rotation matrixes

This chapter describes how other rotation matrixes are calculated. The calculations are not described in the same detail for each matrix as with the Qualisys standard, but you should be able to calculate the exact rotation matrix from these descriptions.

Note: If you use rotations around global axes the order of multiplication of the individual rotation matrixes are reversed and if you use a left-hand system change the positive direction to counterclockwise, which means that the sign of the angle is swapped.

First there are two types of rotation matrixes: those with three different rotation axes and those with the same rotation axis for the first and third rotation.

The first type is of the same type as the one used as Qualisys standard. This means that for this type the same individual rotation matrixes ($\mathbf{R}_x$, $\mathbf{R}_y$ and $\mathbf{R}_z$) are used as for the Qualisys standard. The individual rotation matrixes are then multiplied in different orders to get the different rotation matrix. When you have the rotation matrix the same kind of formulas but with other indexes and signs are used to get the rotational angles as for the Qualisys standard:

$$\text{Pitch} : \phi = \arcsin(r_{13})$$

$$\text{Roll} : \theta = \arccos\left(\frac{r_{33}}{\cos\phi}\right)$$

$$\text{Yaw} : \psi = \arccos\left(\frac{r_{11}}{\cos\phi}\right)$$

You have to look at the rotation matrix to see what indexes and signs that should be used. The singularity will always be at ±90° for the second rotation and at the singularity the third rotation is always set to 0°.

In the second type the third rotation is round the same axis as the first rotation. This means that one of the individual rotation matrixes below is used as the last rotation, depending on which axis that is repeated.

$$\mathbf{R}_{x2} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\theta_2 & -\sin\theta_2 \\ 0 & \sin\theta_2 & \cos\theta_2 \end{bmatrix}$$

$$\mathbf{R}_{y2} = \begin{bmatrix} \cos\phi_2 & 0 & \sin\phi_2 \\ 0 & 1 & 0 \\ -\sin\phi_2 & 0 & \cos\phi_2 \end{bmatrix}$$

$$\mathbf{R}_{z2} = \begin{bmatrix} \cos\psi_2 & -\sin\psi_2 & 0 \\ \sin\psi_2 & \cos\psi_2 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

The rotation matrix (**R**) is then calculated by multiplying two of the individual rotation matrixes from the first type and then one of matrixes above. In the example below the rotations are round the x, y and then x axis again.

$$\mathbf{R} = \mathbf{R}_x \cdot \mathbf{R}_y \cdot \mathbf{R}_{x2} = \begin{bmatrix} r_{11} & r_{12} & r_{13} \\ r_{21} & r_{22} & r_{23} \\ r_{31} & r_{32} & r_{33} \end{bmatrix} =$$

$$= \begin{bmatrix} \cos\phi & \sin\theta_2 \cdot \sin\phi & \cos\theta_2 \cdot \sin\phi \\ \sin\theta \cdot \sin\phi & \cos\theta \cdot \cos\theta_2 - \cos\phi \cdot \sin\theta \cdot \sin\theta_2 & -\cos\theta_2 \cdot \cos\phi \cdot \sin\theta - \cos\theta \cdot \sin\theta_2 \\ -\cos\theta \cdot \sin\phi & \cos\theta_2 \cdot \sin\theta + \cos\theta \cdot \cos\phi \cdot \sin\theta_2 & \cos\theta \cdot \cos\theta_2 \cdot \cos\phi - \sin\theta \cdot \sin\theta_2 \end{bmatrix}$$

The rotation angles can then be calculated according to the equations below. These equations are similar for other rotation matrixes of this type, just the indexes and types are changed.

$$\text{Pitch} : \phi = \arccos\left(\sqrt{1 - r_{11}^{\ 2}}\right)$$

$$\text{Roll} : \theta = \arctan\left(\frac{r_{21}}{-r_{31}}\right)$$

$$\text{Yaw} : \psi = \arctan\left(\frac{r_{12}}{r_{13}}\right)$$

As for the first type of rotation matrixes there will be a singularity for the second rotation angel, but now it is at 0° and 180°. The third rotation angle is then set to 0° and the following rotation matrix can be used to calculate the rotation. It will of course change for other matrixes but the principle is the same.

$$\begin{matrix} 1 & 0 & 0 \\ 0 & \cos(\theta) & -\sin(\theta) \\ 0 & \sin(\theta) & \cos(\theta) \end{matrix}$$

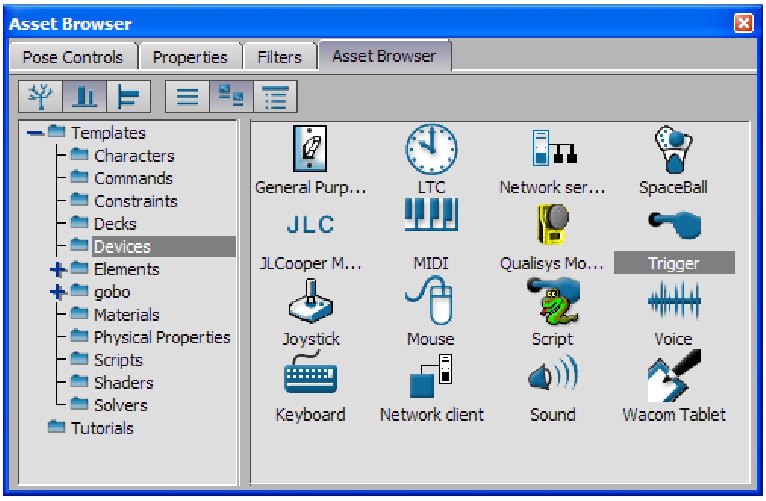
Appendix H: Qualisys MotionBuilder Plugin

How to use Qualisys MotionBuilder Plugin

The Qualisys MotionBuilder Plugin is a device in MotionBuilder that enables real-time streaming of data from QTM. Below there is a description of how to install and use the Qualisys MotionBuilder Plugin. For instructions on how to use the markers in MotionBuilder and other MotionBuilder related questions please refer to the MotionBuilder manual.

1. Run the program Qualisys MotionBuilder plugin.exe to install it in MotionBuilder. It is recommended to have MotionBuilder and QTM installed on different computers for best performance. The plug-in consists of three files: QualisysMocapDevice.dll, devices_qualisys-l.tif and devices_ qualisys.tif, which will be installed in different folders depending on the MotionBuilder version.

   When you start MotionBuilder there should be an icon for the **Qualisys Mocap** device, under **Devices** in the **Asset Browser** in MotionBuilder.



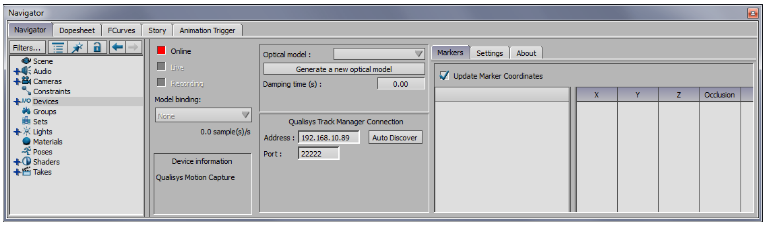
2. Install the MotionBuilder plug-in license in Qualisys Track Manager, see the QTM manual. This is not necessary if you are using Qualisys Track Manager FX.

3. Before starting the Qualisys Mocap device in MotionBuilder make sure that QTM is in RT mode and that you have a identified markers. The RT mode starts with **New** on the **File** menu. To have identified markers activate the **Apply the current AIM models** option on the **Processing** page.

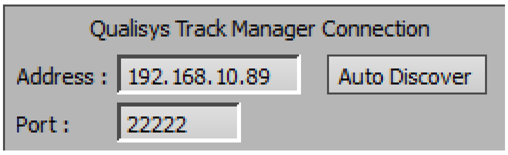
Note: You can also run real time on a file to test the plug-in. Select **Run real time processing on file** on the Capture menu to enable RT output from file.

4. In MotionBuilder drag and drop the **Qualisys Mocap** icon to the main viewer window.

5. You should now see Qualisys Mocap under Devices in the MotionBuilder Navigator window.

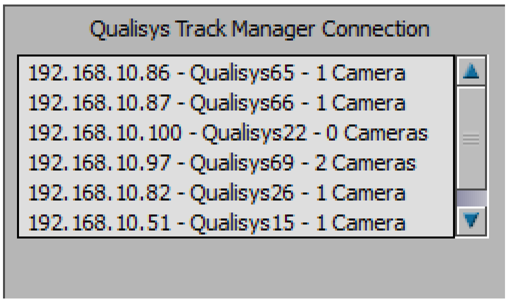


6. Use the **Auto Discover** button to get a list of all the computers on the network that are using QTM.



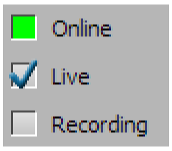
Or enter **Address** and **Port** for QTM computer in the plug-in. Go to the Network connections on the QTM computer to find the IP address. The default port number is 22222 on the QTM RT interface.

Note: The address is 127.0.0.1 or localhost if you are running QTM on the same computer.

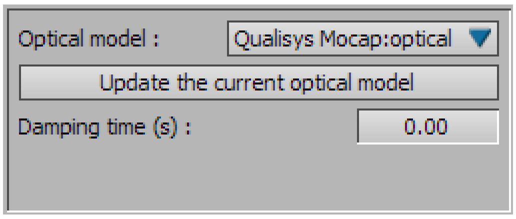


Select the computer that you want to connect to from the list.

7. Press the red **Online** button in the plug-in. If the connection to QTM is established, the **Online** button turns green and the **Live** checkbox underneath is checked.

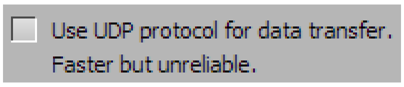


8. Press the **Generate a new optical model** button in the plug-in. Then you should see your markers in the MotionBuilder main viewer window.



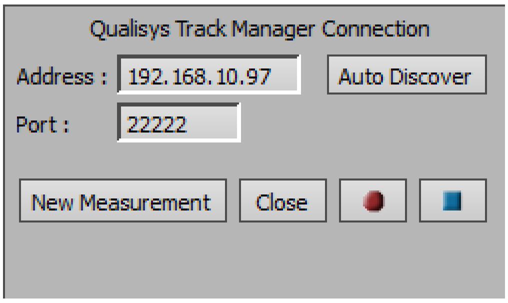
9. There is a **Update Marker Coordinates** checkbox for showing the coordinates of all the markers in the list to the right in the plug-in. This will however slow down the performance of the plug-in.

Click on Settings to the left to show the following two settings.



Use UDP protocol for data transfer
The default setting is to use TCP protocol between QTM and MotionBuilder. In this all packets will reach MotionBuilder but there can be retransmissions which results in a longer delay. With UDP each packet is only sent once and can therefore be lost, this results in a faster data transfer but data can be lost. Each frame of data from QTM is time-stamped so even if one is lost the rest of the data will be placed at the correct time.

10. To record your measurements in MotionBuilder first check the **Recording** checkbox in the Qualisys Mocap device. Then press the record button in the **Transport Controls** window in MotionBuilder. Finally press the play button in the same window. Press stop to end the recording.



With QTM 2.5 and later you can control measurements in QTM from the MotionBuilder plug-in.

New Measurement
Open a new measurement in QTM, i.e. start preview and stream RT data to the plug-in.

Close
Close the current measurement in QTM, i.e. stop streaming RT data to the plug-in.

Capture 
Start a capture in QTM. It will use the capture settings last used by QTM. If you want to change go to QTM and make a measurement with the new settings. A recommended setting is to use **Batch capture**. Then you can trigger the new measurement with the **Capture** button every time you want to start a new.

Stop 
Stop the ongoing measurement in QTM.

Appendix I: QTM iOS apps

Introduction

There are two QTM iOS apps: Viewfinder and Remote. The apps can be downloaded for free from the Apple App store. The iOS apps requires QTM 2.5. Then you also need to be connected to the computer with the camera system. Follow the instructions in the chapters below to use the apps.

The Viewfinder app works on iOS 3.2 or later and can use the higher resolution of iPad and iPhone 4. The Remote app works on iOS 4.0 or later.

Connecting to the QTM computer

For the iOS app to communicate with the QTM computer the iOS device must be connected to the same network as the computer. This can for example either be a network with a wireless router or an ad-hoc network. Check with your system administrator for the best solution.

Network with wireless router
If you already have the network this is the easiest way to connect the iOS app. However if there are a lot of other data sent on the network it may reduce the performance of the iOS app.

Ad-hoc network
For the best performance we recommend an ad-hoc net-work. How to set up an ad-hoc network is described in the chapter “Using an ad-hoc network for the iOS apps” on next page. However this may not be allowed by your system administrator because of security concerns.

To connect to the QTM computer follow these instructions.

1. Connect the iOS device to the wireless network.
2. Start QTM on the computer and make sure that you located the camera system.
3. Start the Viewfinder app and wait for the app to check for QTM hosts.
4. If the iOS device can communicate with the computer you will see it in the list of **Discovered QTM hosts**.

Using an ad-hoc network for the iOS apps

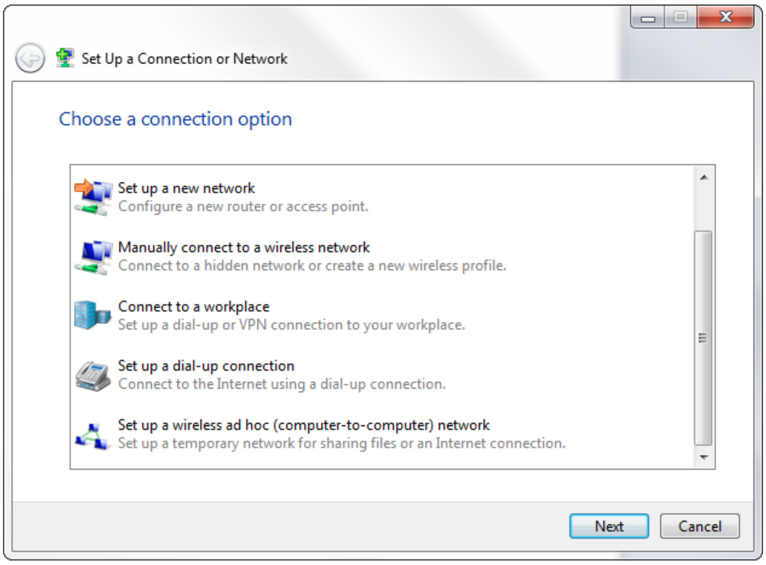
For the best performance of the iPhone apps you have to create an ad-hoc network on the computer with the camera system. Follow the instructions in the following chapters.

Create wireless ad-hoc network on Windows 7

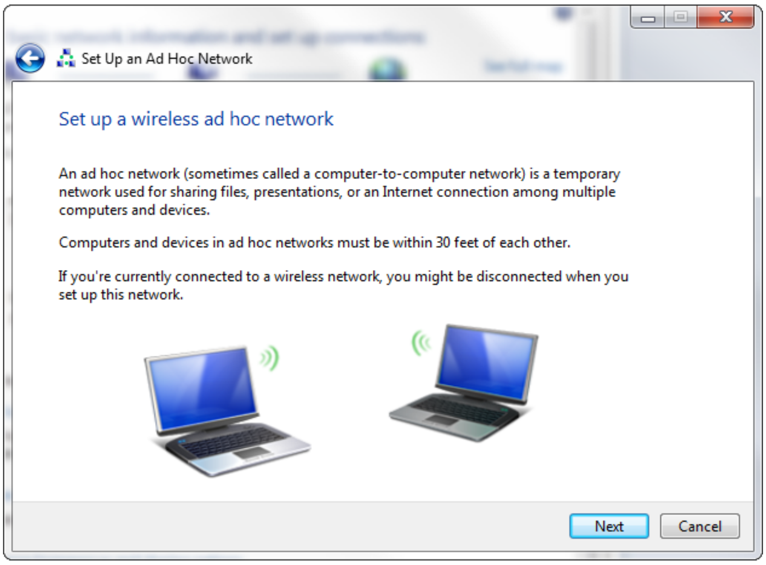
Follow this process to create an ad-hoc network on Windows 7.

Note: Make sure that the driver for the wireless board is updated. There are wireless boards with drivers which do not work with the ad-hoc networks. For example the Dell Latitude E6400 does not work with old drivers.

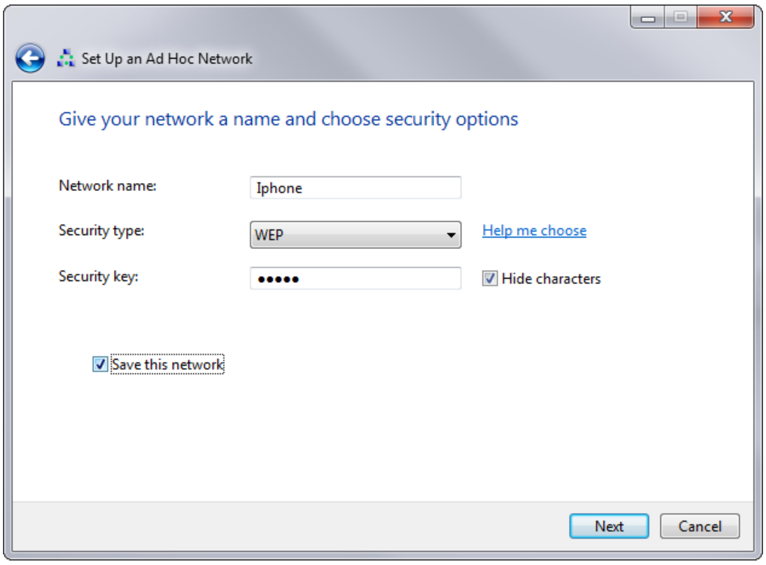
1. Go to the **Network and Sharing Center**.
2. Click on **Set up a new connection or network**.



3. Use the last option **Set up a wireless ad hoc (computer-to-computer) network**. Click on **Next**.



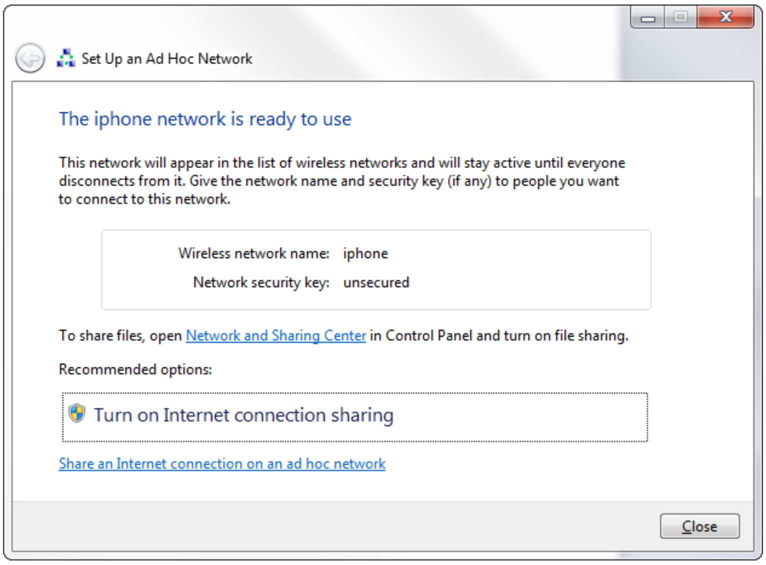
4. Click on **Next**.



5. Enter **Network name** and select **WEP**, enter a **Security key** and **Save this network**. Click on **Next**.

   Note: We recommend using WEP so that there is at least some security used on the network. WPA has better security but it does not seem to work with iOS and an ad-hoc network.

6. Wait for the process to finish.



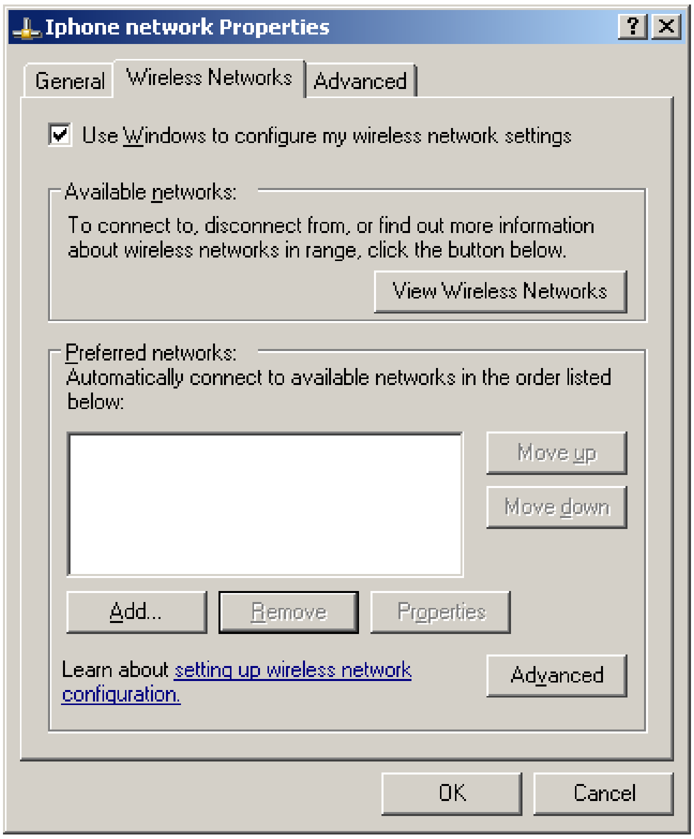
7. Click on **Close.** Do not use the option **Turn on Internet connection sharing** unless you are absolutely sure that it is safe.

8. Continue with configuring the QDS to give the iPhone a IP address automatically, see chapter “QDS settings for the ad-hoc network” on page I - 6.

Create wireless ad-hoc network on Windows XP

Follow this process to create an ad-hoc network on Windows XP.

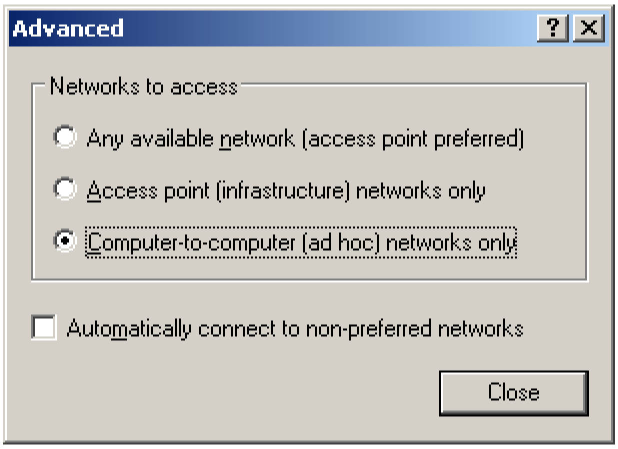
1. Make sure that there is a wireless network on the computer.
2. Open Network connection
3. Right-click on the wireless network and select **Properties**.



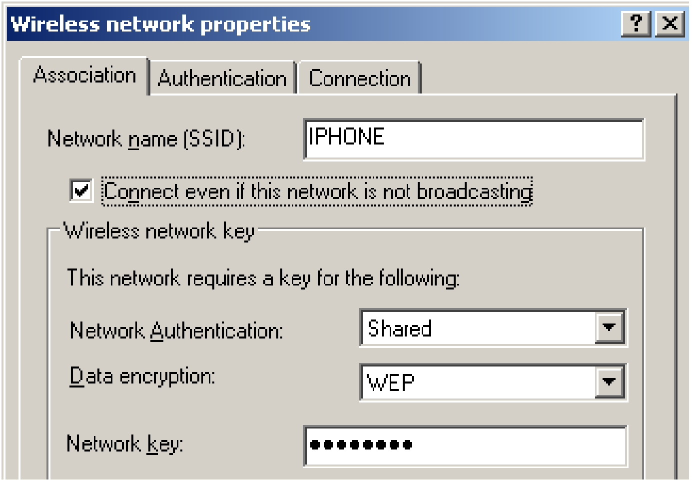
4. Go to the **Wireless Networks** tab.

   a. Select **Use Windows to configure my wireless network settings**, otherwise you need to configure the network via the tool supplied by the manufacturer.

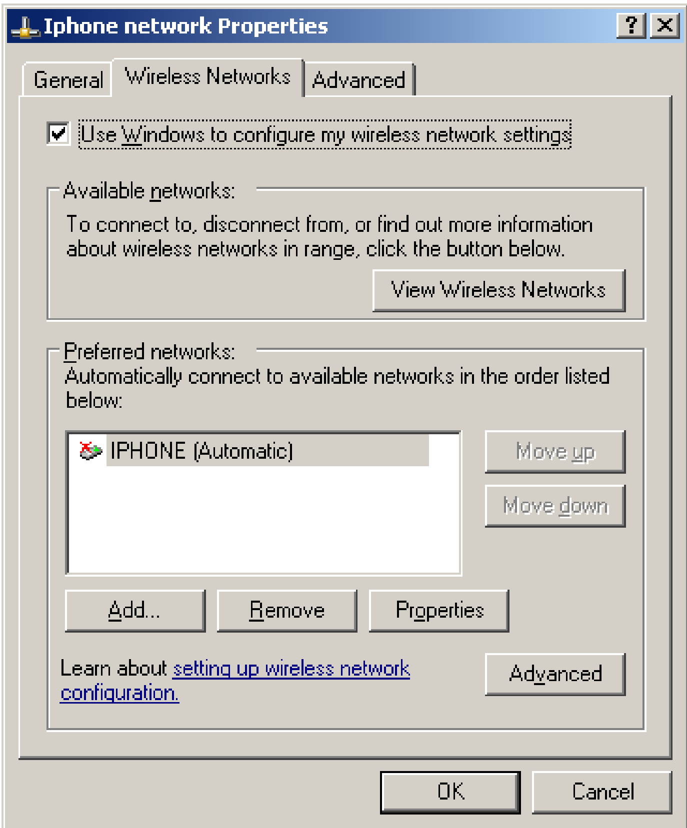
   b. Click on **Advanced** and select **Computer-to-computer (ad-hoc) networks only**. Close the dialog.



5. Click on **Add** and then enter the **Network name**.



a. Select **Connect even if the network is not broadcasting** and then **Shared** for **Network Authentication** and **WEP** for **Data Encryption**. Enter a **Network key** to increase the security. Click on OK and you have created an ad-hoc network.
Note: We recommend using WEP so that there is at least some security used on the network. WPA has better security but it does not seem to work with iOS and an ad-hoc network.



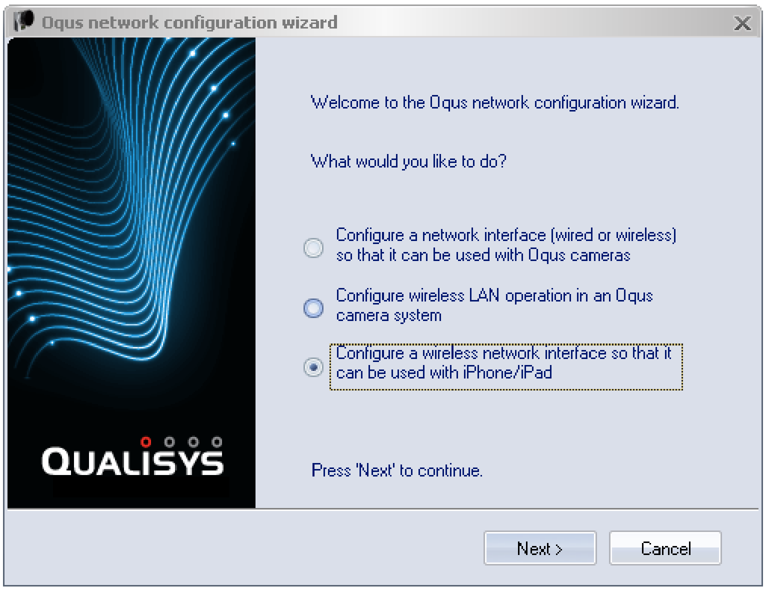
6. Click on OK to close the network properties.
   Note: Do not share your internet connection unless you are absolutely sure that it is safe.

7. Continue with configuring the QDS to give the iPhone a IP address automatically, see chapter “QDS settings for the ad-hoc network” on next page.

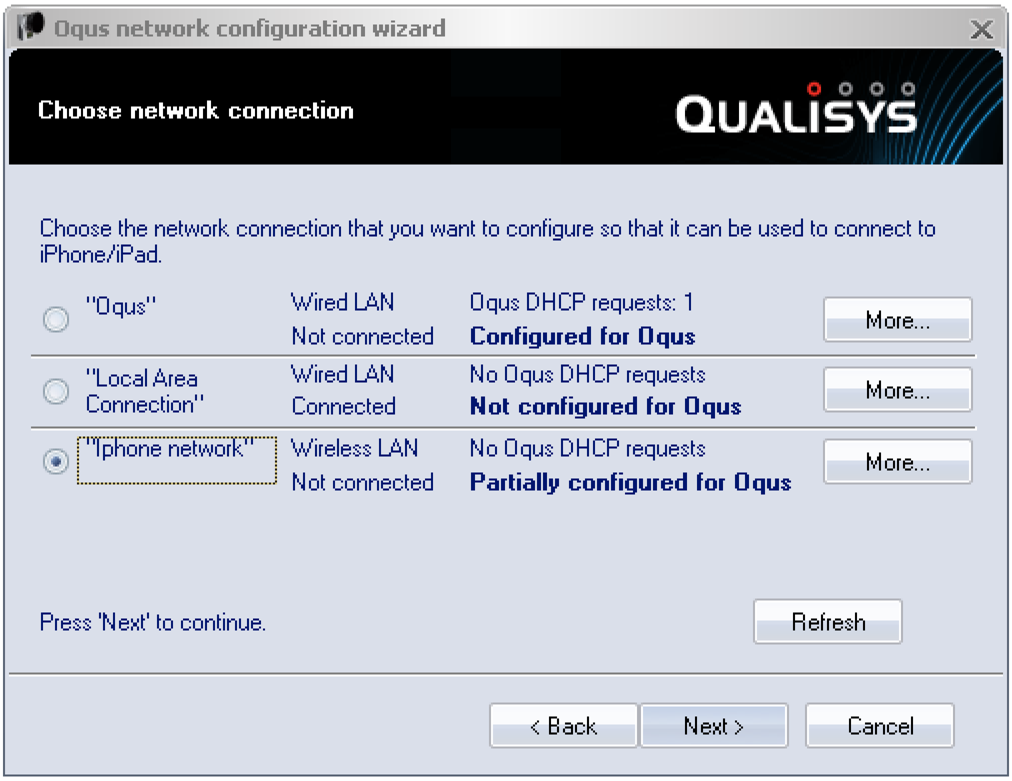
QDS settings for the ad-hoc network

To give iPhones and iPads an IP address automatically you need to configure the QDS. Follow these instructions.

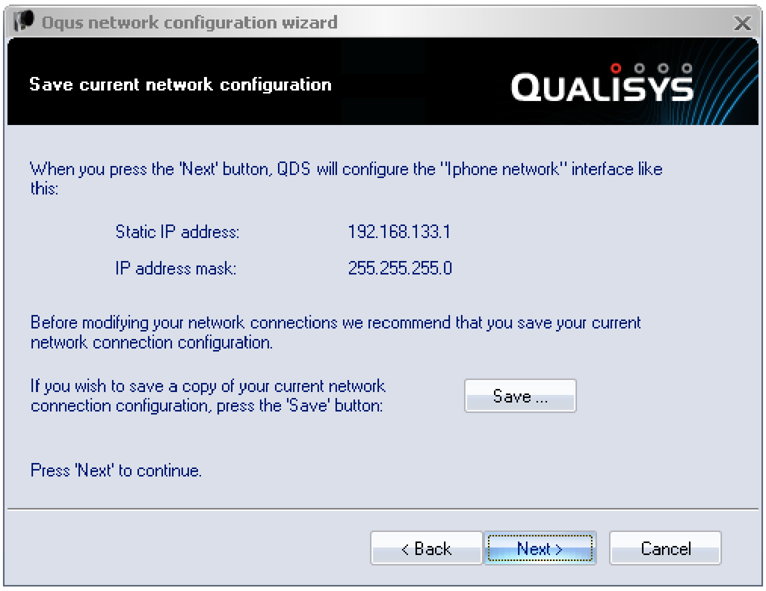
1. Right-click on the QDS in the Windows status bar.
2. Select **Oqus configuration wizard**.



3. Select **Configure a wireless network interface so that it can be used with iPhone/iPad** and click on **Next**.



4. Select the wireless interface that you configured with an ad-hoc network. Then click on **Next**.

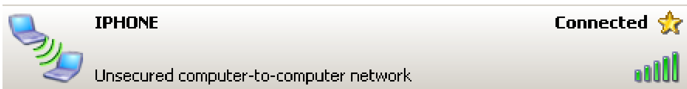


5. The settings that will be used are displayed. The network must IP address 192.168.133.1 and IP address mask 255.255.255.0. Click on **Next**.

   a. a.You can save the current network configuration with the **Save** button.

6. The QDS will change the network configuration it can take some time. Click on **Finish** when it is done.
   Note: In Windows 7 there is a dialog to give QDS elevation to change the network settings. You must have administrator rights to give the elevation.

7. Go to the **View Available Wireless Networks** dialog in Windows. Make sure that wireless network is connected to the ad-hoc network that you created.
   Note: In Windows 7 the network will have the status **Waiting for users** until someone connects to it then it will change to **Connected**.



8. Then connect the iPhone to the network, see chapter “Connect iPhone to the ad-hoc network” below.

Connect iPhone to the ad-hoc network

If you have set up an ad-hoc network with the correct QDS settings according to the previous chapters, it is simple to connect the iPhone. Follow these instructions on your iPhone.

1. Go to **Settings**.
2. Click on **Wi-Fi**.
3. Select the ad-hoc network.
   Note: It may happen that the iPhone selects another network if the ad-hoc network is lost. If you want to make sure that the iPhone does not connect to another network, you have to disable the **Auto-join** setting and forget any saved networks on the iPhone.
4. Click on the >> and wait for the iPhone to receive an IP address. It must start with 192.168.133. If the address starts with 169.254, then the iPhone has not

received the correct address. Try using the **Renew Lease** button and if that does not work delete the ad-hoc network and create it again.

5. When it does you can start using the iPhone apps.

Master/Slave mode in iOS apps

To use all of the features in the Viewfinder and the Remote app, you must be connected as master to QTM. Only one iOS device or other program that uses the RT output at a time can be master. When the iOS app is master it can change settings and control measurements in QTM. If the iOS app is slave it can only view the data which is available in QTM. For the Remote app this means that it cannot be used at all.

The client control can also be turned off or given a password on the **RT output** page in the **Project options** dialog, see chapter “RT output” on page 203. If the client control is disabled no app can become master. When QTM uses a password it must be entered in the app to become master otherwise the app will be connected as slave.

Viewfinder app

The Viewfinder app can be used to check what the camera system can see, for example when setting up the camera system. With the app you can view the marker, marker intensity and video image of an Oqus camera. On ProReflex cameras you can only view the marker image.

The Viewfinder app has settings in the iOS setttings, see chapter “Viewfinder app settings” on page I - 13.

Connecting the Viewfinder app

To connect the Viewfinder app to the current QTM computer follow these instructions. Only one iOS device at a time can be master and control QTM. If another device connected first or if QTM does not allow you to control the system, you will be slave and can only view the data which is available in QTM, see chapter “Master/Slave mode in iOS apps” above.

1. Start QTM and make sure that you located the camera system.
   Note: Make sure that you only have a 3D window in preview for the best performance in video mode.

2. Start the Viewfinder app.

3. Wait for the app to discover the available hosts. Select the computer you want to connect to from the **Discovered QTM hosts** list and then click **Connect**. You can also enter the IP address in the **QTM host** setting. If a password is needed enter that in the **Password** setting.
   Note: The password is set on the **RT output** page in the **Project options** dialog.
   Note: The last IP number that was used by the app will be entered in the **QTM host** setting. To clear this address go to Viewfinder app settings, see chapter “Viewfinder app settings” on page I - 13.

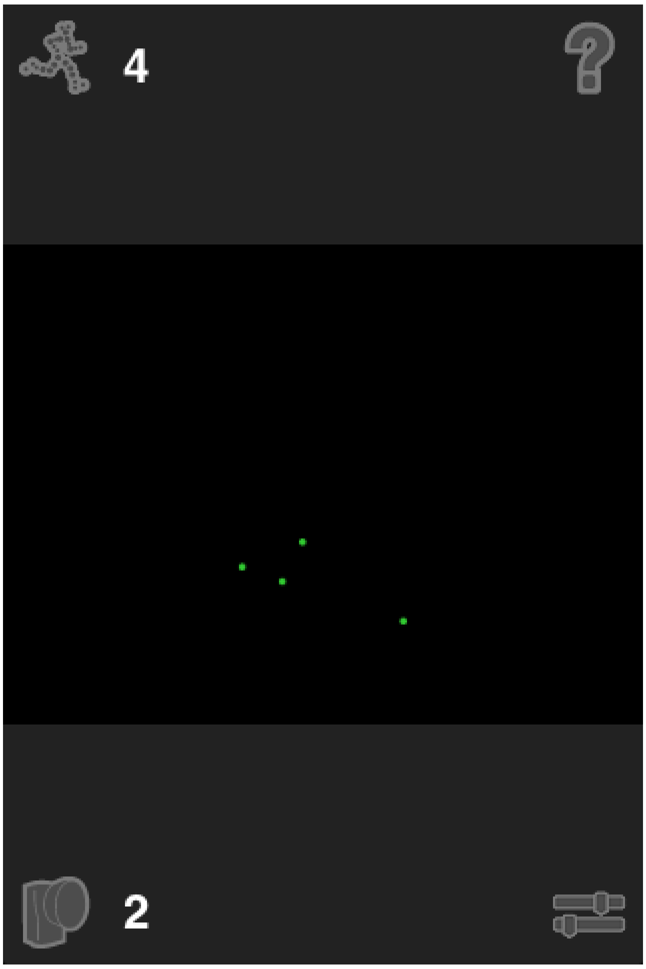
   a. Click on the **Demo** button to view a demo of the marker mode. In the demo you can switch cameras and zoom in on the marker data.

4. Wait for the app to connect to QTM.
    a. If you have not entered the correct password the app will ask for it again. Select **Change** to enter the password again or select **Slave mode** if you just want to look at the images.

5. When the app is connected it will flash whether it is master or slave. For more information about how to use the app see chapter "Using the Viewfinder app" on next page.

Using the Viewfinder app

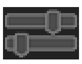
In the Viewfinder app you can view the camera images and change some camera settings. Any changes done in the app or in QTM is displayed immediately in the respective program. This means for example that if you change the exposure time in the app, then the exposure time slider will move in QTM.



The Viewfinder app includes the following options.

The Viewfinder app can view markers, marker intensity image and video image from the cameras. Select the mode by clicking on the icon in the top left corner. When you switch mode all cameras will switch mode in QTM. Therefore you get the best performance in marker intensity and video mode if you only have a 3D window open.
The app will circle through the modes in the order shown above. In marker mode you can see the number of markers next to the icon.
For information on how to modify the image with gestures see chapter “Gestures in the Viewfinder app” on next page.

The camera number is displayed next to the camera icon in the bottom right corner. Change the camera either by swiping left or right in the image or by clicking on the camera icon and moving the slider to the camera you want.



Click on the icon in the bottom right corner to open the camera settings. You can change **Exposure & flash time** and **Marker threshold** settings in marker mode. In Video mode you can change the **Exposure time** and **Flash time** settings.

Note: The settings are only changed for the camera that is currently displayed in the app.

Use the **About** button for information about the app and how to use it.

Gestures in the Viewfinder app

Use the following gestures to change the image in the Viewfinder app. The gestures works on the image even if the camera settings are displayed.

Swipe left/right
Change camera that is displayed. The number of the camera flashes when you switch.

Pinch
Zoom in continuously on the image.

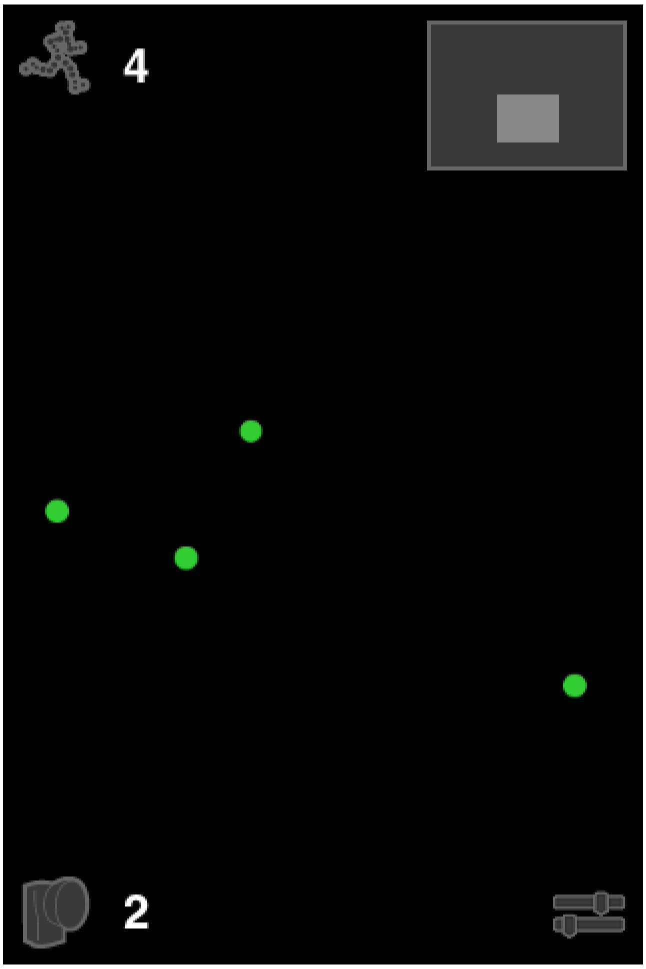
Drag
Pan the zoomed image.

Double tap
Zoom in using steps

Two finger tap
Zoom out using steps

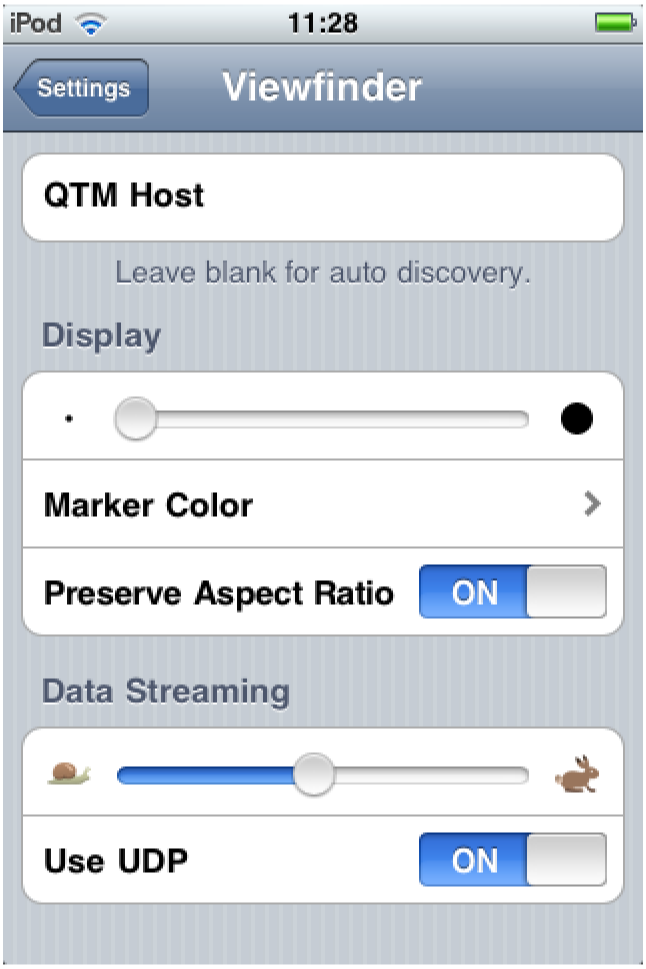
Three finger tap
Reset the zoom to full field of view.

When the image is zoomed in, a square in the upper right corner displays how much and where you have zoomed. Double-click in that square to reset the zoom to full field of view.



Viewfinder app settings

There are settings for the Viewfinder app on the iOS setting. Click on **Settings** and then scroll down to the Viewfinder app.



You can change the following settings:

QTM Host
Set the default IP address in the Viewfinder app. This is changed to the last IP address used in the Viewfinder app.

Display
Change the following display settings

Marker size
Drag the slider to change the marker size.

Marker color
Set the color of the markers.

Preserve Aspect Ratio
The default is **ON**, which means that the aspect ratio of the current camera is kept in the app. Turn **OFF** to fill as much the screen as possible.

Data Streaming
Change the following data streaming settings.

Speed
Set the speed of the Viewfinder app, i.e. how often it asks QTM for an update. This usually does not have to be changed. However you can decrease the speed if you have a bad connection or increase it with a good connection.

Use UDP
The default is ON because it gives the highest speed. However if the app

does not receive any frames from QTM, try turning it off. Because some firewalls and routers may block the UDP packages.

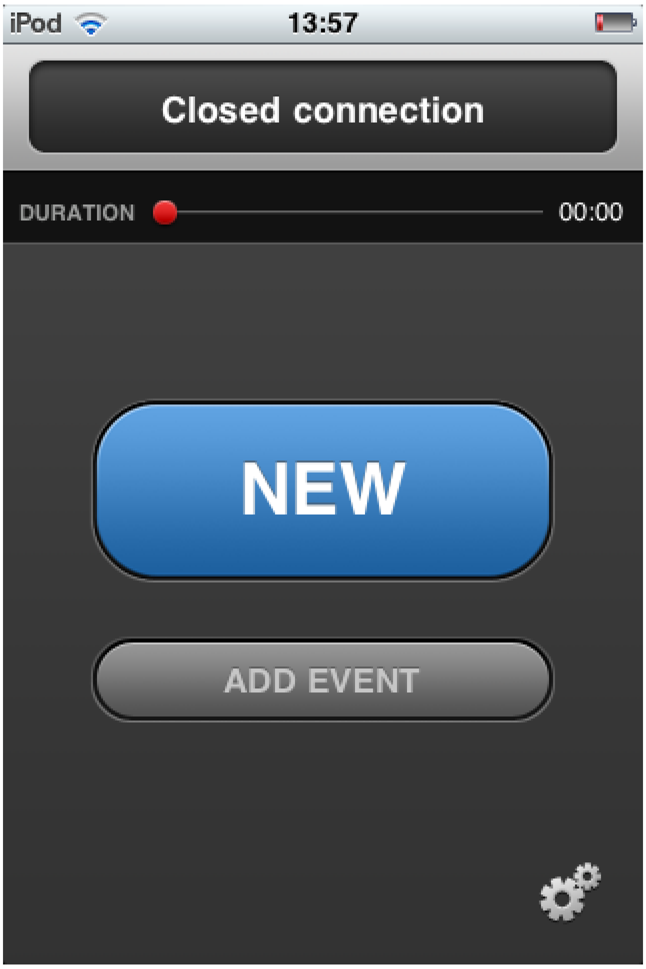
Remote app

The Remote app is used to control the start and stop of the measurements in QTM. The app can also change the measurement time and set events during a measurement. The app works with both Oqus and ProReflex cameras.

Connecting the Remote app

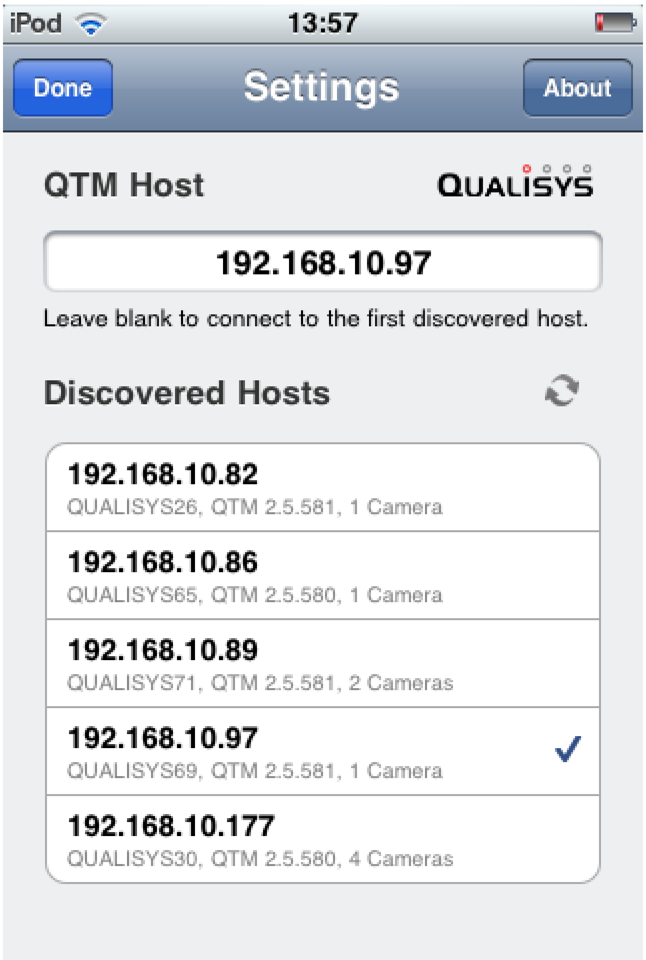
To connect the Remote app to the current QTM computer follow these instructions. Only one iOS device at a time can be master and control QTM. If another device connected first or if QTM does not allow you to control the system, then you cannot use the Remote app.

1. Start QTM and make sure that you located the camera system.
2. Start the Remote app.
    a. The first time you start the app it will try to connect to the first computer in the **Discovered Hosts** list.



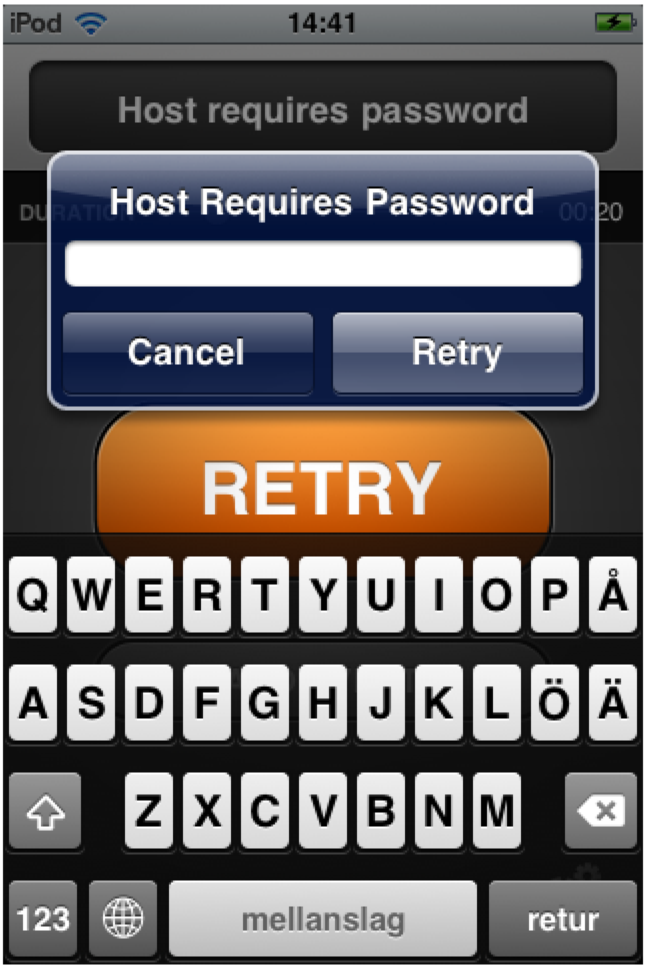
3. Click on the settings button  if you want to specify the computer you are connected to.

4. Wait for the app to discover the available hosts. Select the computer that you want to connect to from **Discovered hosts** list. Update the list with the button. You can also enter the IP number directly in the **QTM Host** option.



5. Then click on **Done**.

6. If QTM requires a password enter it in the dialog.
   Note: The password is set on the **RT output** page in the **Project options** dialog.



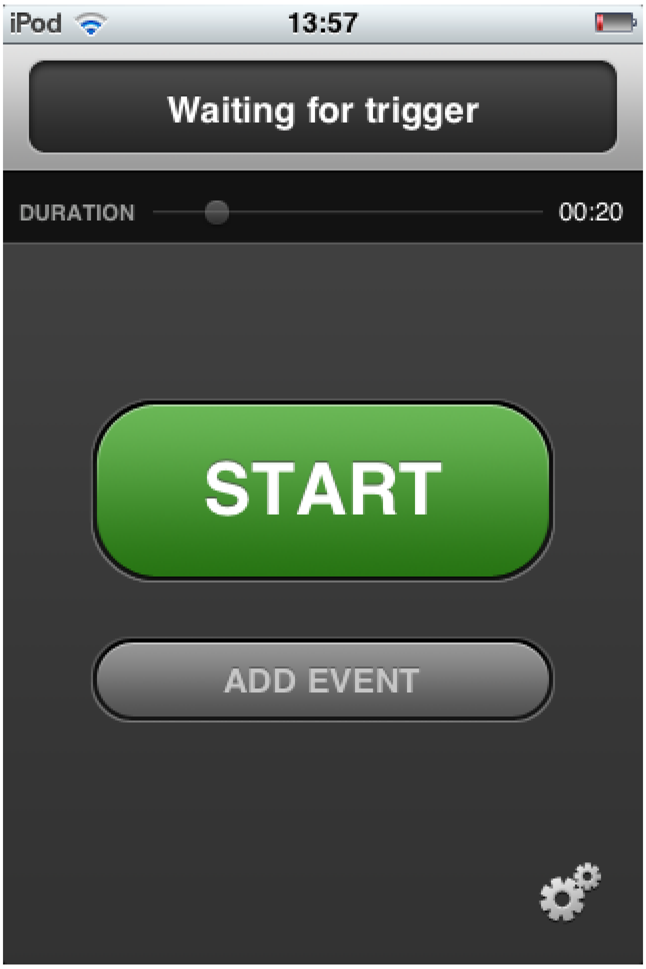
Using the Remote app

The Remote app can control start, stop and events in QTM. There are some different ways to control the measurement and these are described below.

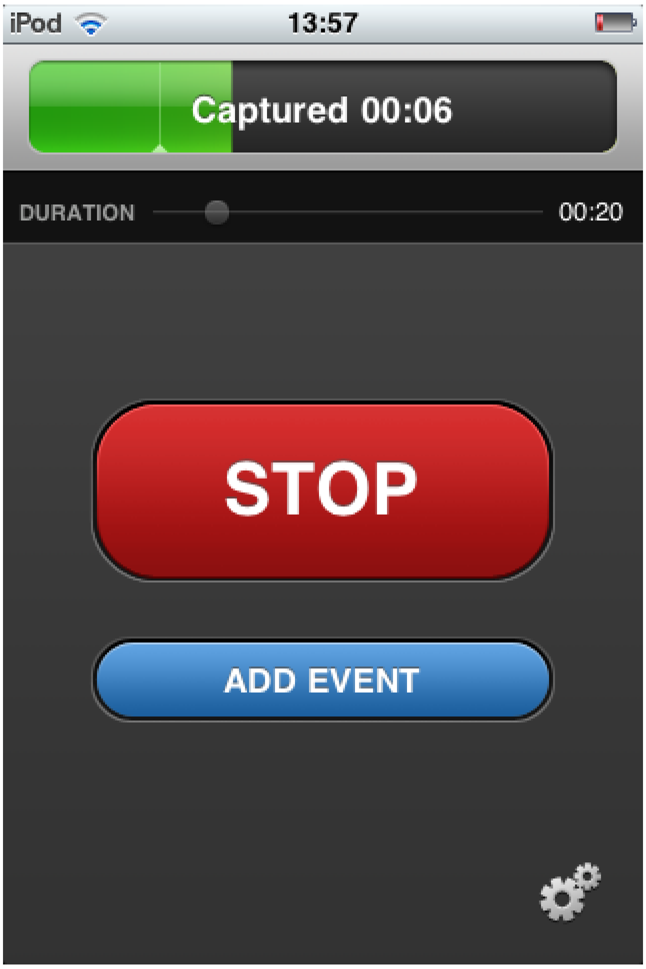
⚠ IMPORTANT: Because it needs to use external trigger when starting a measurement that option is activated automatically in QTM as soon as the Remote app connects.

Remote app as external trigger

The Remote app will detect the status of the measurement in QTM. Therefore if you start the measurement on the computer and QTM is waiting for trigger that will be reported in the Remote app. Then you can use the **Start** button as an external trigger button.



During the measurement you can use the **Add event** button to set events. The events are marked with a thin line in the measurement progress of the Remote app. You can also stop the measurement with the **Stop** button.



Making measurements from the Remote app

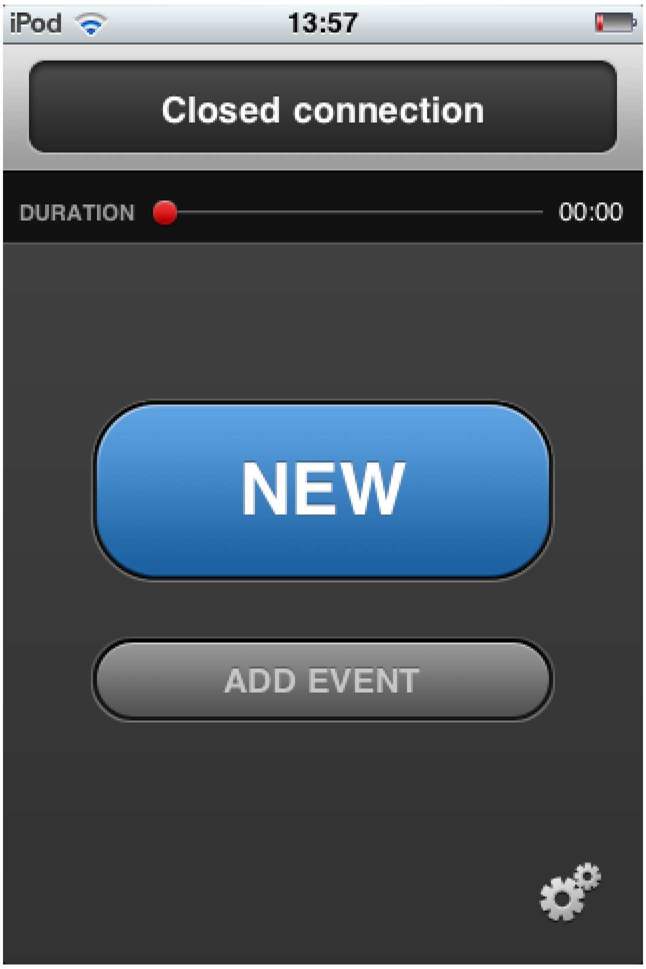
You can use the Remote app to make measurements in QTM. Follow these steps:

1. Make sure that all the settings are correct in QTM.

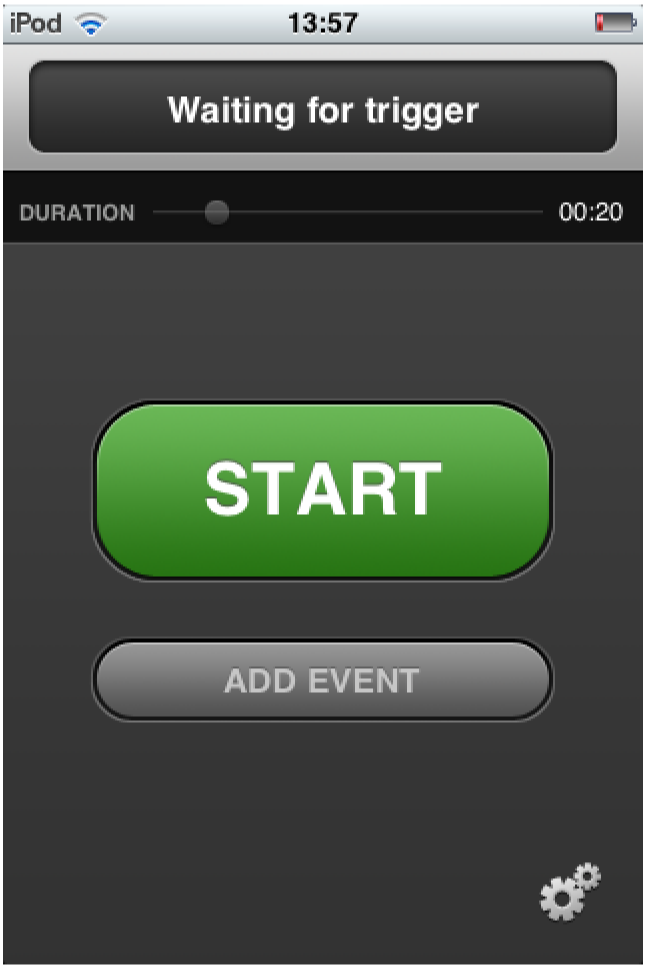
    a. The only setting you can change from the Remote app is the measurement time. However if you do not change the setting in the Remote app then it will receive the time from QTM.

    b. If you want to save the file automatically you have to open the **Start capture** dialog in QTM. Open the dialog with the **Capture** button and then specify the settings for the **Automatic capture control**.
    Note: You do not have to use Batch capture, because you can start a new measurement from the Remote app. However if you use Batch capture then you only have to press the button in the Remote app once between the measurements.

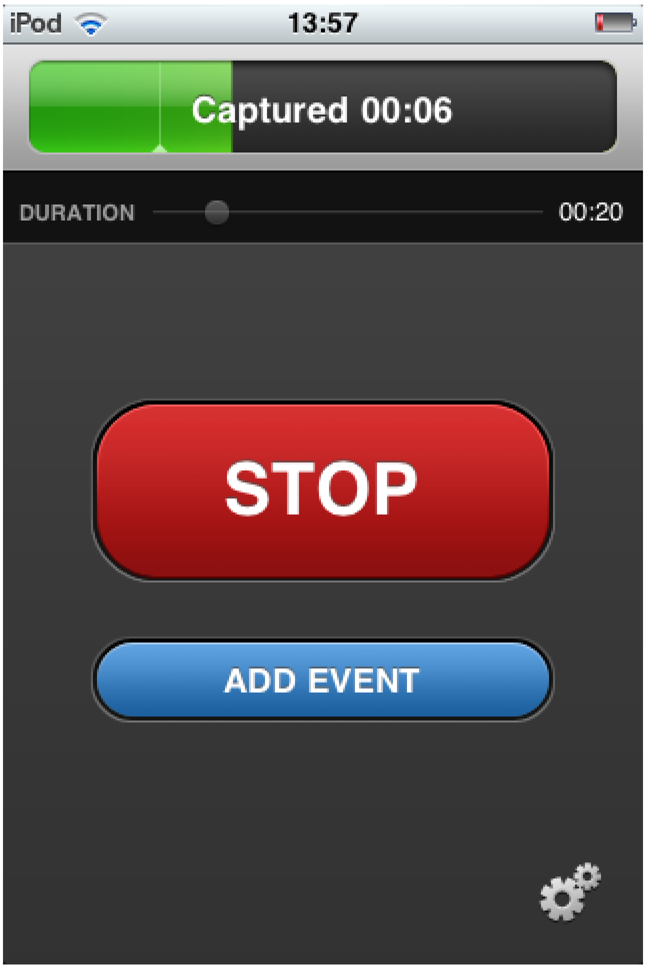
2. Click on **New** in the Remote app. This is the same as pressing the **Capture** button in QTM. However there will be no **Start capture** dialog.
    Note: The currently open file will be closed in QTM. If the file has unsaved changes QTM will open a new instance.



3. Wait for the app to report **Waiting for trigger**.



4. Press the **Start** button in the Remote app when you want to start the measurement.



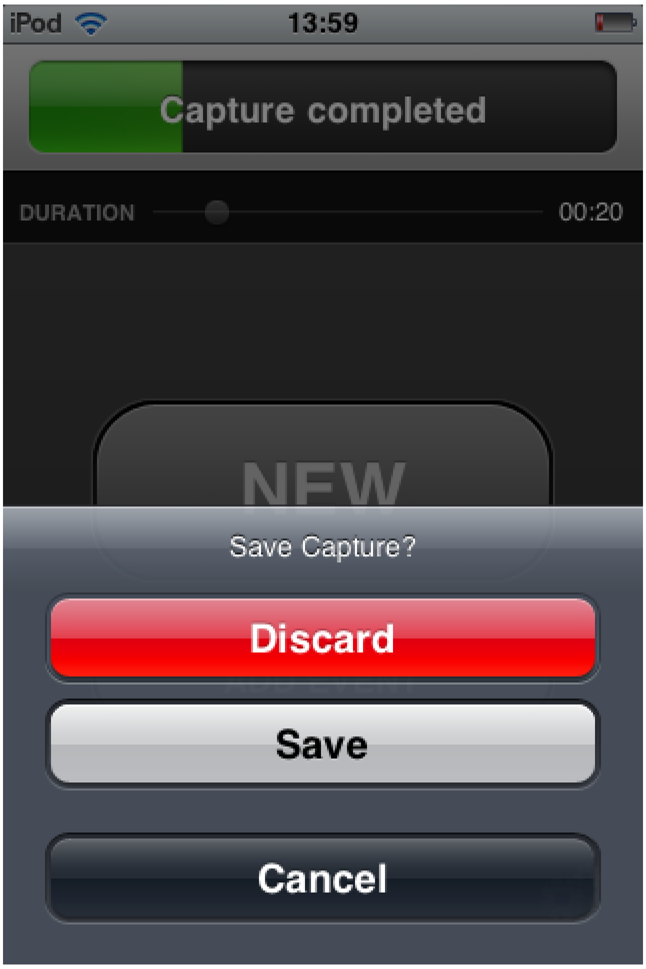
5. During the measurement you can add events with the **Add event** button, which is displayed as a thin line in the measurement progress bar. You can also stop the measurement with the Stop button.

6. After the measurement is finished the status of the app depends on whether batch capturing and saving is turned on or not.

   Batch capture
   When using batch capture the Remote app will return to the **Waiting for trigger** status after a measurement and you can trigger the next measurement. If the app does not return to the **Waiting for trigger** state, check if there is any message in QTM.
   To stop making measurements you have to end it on the computer.

   Only saving file automatically
   In this mode the file is saved automatically after the measurement and then displayed in QTM. The app will return to the **Closed connection** state. Repeat the steps 2-4 to perform a new measurement.
   Note: The file currently open in QTM will always close when you start a new measurement. The setting for **Measurement file close** on the **3D view settings** page does not change this behavior.

   Not saving file
   In this mode the file is displayed in QTM and the app returns to the **Closed connection** state. When you press the **New** button the app will ask if you want to **Save** or **Discard** the file. Repeat the steps 2-4 to perform a new measurement.


iPod
13:59
Capture completed
DURATION
00:20
NEW
Save Capture?
Discard
Save
Cancel

Appendix J: EU customer information

Waste Electrical and Electronic Equipment (WEEE)

In the European Union (EU), waste from electrical and electronic equipment (WEEE) is now subject to regulation designed to prevent the disposal of such waste and to encourage prior treatment measures to minimize the amount of waste ultimately disposed. In particular, the EU WEEE Directive 2002/96/EC requires that producers of electronic equipment be responsible for the collection, reuse, recycling and treatment of WEEE which the producer places on the EU market after August 13, 2005. Qualisys is providing the following collection process to comply with the WEEE Directive.

Qualisys WEEE Collection Process

If you have purchased Qualisys products in the EU on and after August 13, 2005, and are intending to discard these products at the end of their useful life, please do not dispose of them in a landfill or with household or municipal waste. Qualisys has labeled its electronic products with the WEEE label to alert our customers that products bearing this label should not be disposed with waste in the EU. Instead, Qualisys requests you to return those products using the instructions provided here, so that the products can be collected, dismantled for reuse and recycled, and properly disposed.

Qualisys will take back WEEE, i.e. all of the electrical equipment which is part of Qualisys equipment, from its customers within the EU. Please visit the website www.qualisys.com/weee or contact Qualisys AB at weee@qualisys.com for information on how to return your WEEE.

Appendix K: China ROHS

有害物质声明

按照中华人民共和国电子工业标准SJ/T11364 2006的要求，此文档提供了由Qualisys AB生产制造的Oqus1-，3-和5-系列的危险材料声明。

部件名称	有毒有害物质或元素					
	铅	汞	镉	六价铬	多溴联苯	多溴二苯醚
印制电路配件	x	o	o	o	o	o
显示器	x	o	o	o	o	o
按钮	o	o	o	o	o	o
内部配线	o	o	o	o	o	o
围栏	o	o	o	o	o	o
镜头	x	o	o	o	o	o
外接电缆及端口	o	o	o	o	o	o
交流电/直流电电源	o	o	o	o	o	o
纸质说明书	o	o	o	o	o	o
CD说明书	o	o	o	o	o	o

O:表示该有毒有害物质在该部件所有均质材料中的含量均在SJ/T 11363-2006标准规定的限量要求以下。

X:表示该有毒有害物质至少在该部件的某一均质材料中的含量超出SJ/T 11363-2006标准规定的限量要求。

Revision history

QTM 2.6

The following major changes have been added to the manual for QTM 2.5 (build 613).

New features and changes

1. The measurements in QTM are organized in projects, see chapter “Projects” on page 8 and “Using projects” on page 8.
    a. The QTM startup procedure, see chapter “QTM startup procedure” on page 9.
    b. Create a new project, see chapter “Creating a new project” on page 10.
    c. Switching projects, see chapter “Switch project” on page 11.
2. QTM support for the 4-series camera, see applicable chapters in “Project options” on page 107 and “Oqus - Camera manual” on page A - 1.
3. Options to control the location of files, see chapter “Folder options” on page 224.
4. Options for the startup procedure of QTM, see chapter “Startup” on page 225.
5. Display number of trajectories in the current frame in the 3D view, see chapter “3D view settings” on page 221.

QTM 2.5 (build 613)

The following major changes have been added to the manual for QTM 2.5 (build 613).

New features and changes

1. Oqus 3+ information.
    a. Comparison with the other camera models, see chapter “Oqus 3+ features compared to other Oqus models” on page 345.
    b. Sensor mode setting for higher frame rate at full FOV, see chapter “Sensor mode” on page 155.
2. Active filtering for the Oqus 3+ and 1-series, see chapter “How to use active filtering” on page 264.
3. New 6DOF features and changes.
    a. Virtual markers from 6DOF data, see chapter “Virtual markers calculated from 6DOF data” on page 292.
    b. Naming the points in the 6DOF body, see chapter “Definition of 6DOF bodies” on page 287.

c. Define 6DOF body saves the body in both Project option and the current file, see chapter “Definition of 6DOF bodies” on page 287.

d. Changes in how the 6DOF data is handled in reprocessing, see chapter “Calculating 6DOF data” on page 290.

e. Changed the rotation option **Define x y plane with 3 points in body**, see chapter “Rotate body” on page 183.

f. Added angles to the **Calculate** option for 6DOF bodies, see chapter “Calculate” on page 84.

4. Filter data in plots from the **Data info** window, see chapter “Data types” on page 80.
5. Changed so that corrected merged markers are always used in the 3D tracker, see chapter “How to use circularity filtering and correction of merged markers” on page 267.
6. Linearization file warning if there is mismatch between camera and QTM, see chapter “Linearization file warning” on page 234.
7. Warning before calibration with delayed exposure, see chapter “Delayed exposure to reduce reflections from other cameras” on page 252.
8. Auto-backup of files in batch processing, see chapter “Batch processing” on page 279.
9. Gap-fill setting saved in file so that it can be used in reprocessing, see chapter “Reprocessing a file” on page 281.
10. Use individual marker size settings for each file, see chapter “3D view settings” on page 221.
11. New angle calculations in Analyze, angular acceleration and calculate angle from two markers, see chapter “Calculation” on page 75.
12. Calculate **Distance traveled** of a trajectory, see chapter “Calculation” on page 75.
13. Delete imported QVA 2D data, see chapter “Analyzing data in video files” on page 261.
14. No automatic zeroing of the force data in files on new versions of the portable AMTI plates, see chapter “AMTI 8-Channel force plate calibration parameters” on page 198.

New information

15. Use multiple files in AIM models for best performance, see chapter “Generating an AIM model” on page 302.
16. How to use 6DOF in different setups, see chapter “How to use 6DOF bodies” on page 296 and “How to use 6DOF bodies in an AIM model” on page 296.
17. Virtual markers in AIM models, see chapter “How to use 6DOF bodies in an AIM model” on page 296.
18. Use marker filtering to improve 6DOF data, see chapter “How to improve 6DOF data with marker filtering” on page 297.
19. Information about loading workspace and calibrations, see chapters “Loading an old workspace file” on page 277 and “Loading a saved calibration” on page 277.

QTM 2.5

The following major changes have been added to the manual for QTM 2.5.

New features and changes

1. QTM iOS apps for Remote control and Viewfinder, see “QTM iOS apps” on page I - 1.
2. Automatic detection of need to calibrate, see chapter “Calibration quality” on page 127.
3. Improved marker filtering and merged marker correction, see chapter “How to use circularity filtering and correction of merged markers” on page 267 and “Marker circularity filtering and correction of merged markers” on page 148.
4. New options on the 2D view window menu.
    a. Rotate camera view, see chapter “2D view window menu” on page 44.
    b. Select Oqus compression, see chapter “2D view window menu” on page 44.
    c. Review camera settings in files, see chapter “Review camera settings” on page 47.
5. Wizard for graphic board problems, see chapter “Hardware requirements” on page 4.
6. Delayed exposure setting displayed in 2D views, see chapter “Delayed exposure to reduce reflections from other cameras” on page 252.
7. Setting for excluding force and analog data in TSV export, see chapter “Data type to export” on page 208.
8. Export SMPTE timecode in TSV export, see chapter “General export settings” on page 208.
9. Events and SMPTE timecode included in the MAT file export, see chapter “Matlab file export” on page 212.
10. Format of the direct to Matlab export is now the same as the MAT file export, see chapter “Export directly into Matlab” on page 326.
11. New video format for Point Grey cameras, see chapter “Capturing video with Point Grey” on page 369.
12. Setting to allow client control in RT, see chapter “Real time server communication” on page 203.
13. New visualization during the linearization, see chapter “Linearization instructions” on page 230.
14. Setting for opening the Qualisys working folder, see chapter “Project folder” on page 12.
15. Show bounding box in 3D view, see chapter “3D view settings” on page 221.
16. COP threshold uses only the vertical force of the force plate coordinate system, see chapter “COP (Center Of Pressure) threshold” on page 202.
17. Option for rotating the force plate location 180 degrees, see chapter “Force plate location” on page 199.

18. New MotionBuilder plugin, see chapter “How to use Qualisys MotionBuilder Plugin” on page H - 1.
    a. Auto discover QTM computer.
    b. Control measurements in QTM.
    c. Auto-detect vertical coordinate axis.

New information

19. New installation instructions for Point Grey cameras, see chapter “Installing the Point Grey camera” on page 367.
20. Updated hardware and plug-in requirements, see chapters “Hardware requirements” on page 4 and “Plugin requirements” on page 5.
    a. The analog board PCI-DAS6402/16 is now working on Windows 7.
    b. Vista no longer tested in QTM 2.5, because of the inferior performance of that OS.
21. Advice on calibration with delayed exposure, see chapter “Delayed exposure to reduce reflections from other cameras” on page 252.
22. More information about the Calibration quality check, see chapter “Calibration quality warning” on page 240.
23. More information about the analog offset warning, see chapter “Analog offset warning” on page 357.
24. Updated descriptions of Noraxon connections, see chapter “Noraxon EMG” on page F - 1.
25. Analog boards PCI-DAS1602/16, PCIM-DAS1602/16 and PC-CARD DAS16/16 are no longer tested in QTM, see chapter “Old analog hardware” on page E - 2.

QTM 2.4

The following major changes have been added to the manual for QTM 2.4.

1. Description of how to use manual events, see chapter “Events” on page 309.
2. Information about how to use multiple files for the AIM model generation, see chapter “Generating an AIM model” on page 302.
3. Updated requirements for Windows 7, see chapter “Hardware requirements” on page 4.
4. New sound and OSC related functionality.
    a. Description of how to use the box for SMPTE timestamp, see chapter “Using synchronization box for SMPTE” on page 384.
    b. New settings for the external timebase, see chapter “External timebase” on page 135.
    c. Information about the OSC integration, see chapter “OSC integration” on page 276.
5. Point Grey integration, see chapter “How to use Point Grey cameras” on page 367.
6. New recommended codec called MJPEG, see chapter “Codecs for Oqus high-speed video files” on page 261 and “FFDS-MJPEG codec settings” on page 154.

7. New Noraxon integration, see chapter “Noraxon EMG” on page F - 1.
8. Export of force data in TSV files, see chapter “General export settings” on page 208 and “Force data (_f.tsv)” on page 317.
9. New 6DOF settings
    a. New setting for the real-time 6DOF tracker, see chapter “6DOF Tracker parameters” on page 179.
    b. Added buttons for color and coordinate system settings, see chapter “Rigid bodies” on page 180.
10. Description of how to use an ethernet switch with the Oqus system, see chapter “Setup Oqus system with Ethernet switch” on page 342.
11. Changes to analog offset and drift.
    a. New settings, see chapter “Compensate for analog offset and drift” on page 163.
    b. Analog offset and drift can now be toggled in a captured file, see chapter “Analog data information” on page 82.
12. On new installations the files generated by QTM such as calibrations are saved in a new location, see chapter “Project folder” on page 12.
13. New toolbar button for analyze trajectory, see chapter “AIM toolbar” on page 99.
14. Information on how to set the jumpers on the Sealevel 7106 board, see chapter “Sealevel Low Profile PCI ULTRA 530.LPCI [7106S-SN]” on page C - 3.
15. New setting for the audio playback of DV video, see chapter “2D view settings” on page 220.
16. Settings for control of the OpenGL rendering, which can help on computers with integrated graphic boards, see chapter “3D view settings” on page 221.
17. Select components in installation, such as Noraxon EMG driver and Mega EMG driver, see chapter “Software installation” on page 6.
18. New 3D tracking parameter for minimum trajectory length, see chapter “Minimum trajectory length” on page 173.
19. Select range for the Euler angles, see chapter “Definition of rotation axes” on page 188.
20. Enter custom values for the Gain and Excitation voltage of AMTI force plates, see chapter “AMTI force plate settings” on page 192.
21. Skip gap option in the **Gap fill trajectory** dialog, see chapter “Gap fill trajectory with preview” on page 73.

QTM 2.3

The following major changes have been added to the manual for QTM 2.3.

1. Display of covered and calibrated volumes in 3D view windows, see chapter “Volumes in 3D views” on page 57.
2. Display of camera view cones in 3D view windows, see chapter “Camera view cones in 3D views” on page 56.

3. DV and webcam video moved to the 2D view window, see chapter "DV/webcam video in 2D view" on page 42.
   a. Compression for DV and webcam avi-files, see chapter "Compression of video from video cameras" on page 365.
   b. Zoom functionality added to the DV video image, see chapter "2D view window" on page 36.
4. Display of real-time latency added for Oqus cameras, see chapter "Real time latency" on page 275.
5. Column headers and time/frames added to TSV format, see chapter "General export settings" on page 208 and "TSV file formats" on page 315.
6. QTM checks for update at startup, see chapter "Software installation" on page 6.
7. New settings on Advanced timing for Oqus cameras.
   a. Changed settings for external timebase, see chapter "External timebase" on page 135.
   b. Setting called **Measurement time** added to synchronization output, see chapter "Synchronization output" on page 137.
8. Reduce reflections with delayed exposure, see chapter "Delayed exposure to reduce reflections from other cameras" on page 252.
9. New setting for display of plot toolbar, see chapter "GUI" on page 218.
10. New setting to reset hidden message boxes, see chapter "GUI" on page 218.

QTM 2.2

The following major changes have been added to the manual for QTM 2.2.

1. New and improved 3D GUI.
   a. Settings for the objects in the 3D view, see chapter "3D view settings" on page 221.
   b. Toolbar for tools in the 3D view, for example for **Zoom** and **Quick identification**, see chapter "3D view toolbar" on page 52.
   c. Drag and drop of markers in the 3D view to Trajectory info windows, see chapter "Trajectories in 3D views" on page 54.
   d. Tool tips for markers, force plates and cameras, see chapters "Trajectories in 3D views" on page 54 and "Other objects in 3D views" on page 60.
   e. Force trace (Pedotti diagrams) for the force data, see chapter "Viewing force data" on page 311.
   f. Cut marker traces with the **Cut trajectory trace** tool, see chapter "3D view toolbar" on page 52.
   g. Modified **3D view window** menu, with for example **Reset viewpoint**, "3D view window menu" on page 60.
2. New and improved 2D GUI.
   a. **Camera settings** sidebar for the basic Oqus camera settings, see chapter "Camera settings sidebar" on page 40.

b. Toolbar for tools in the 3D view. For Oqus there are tools for **Image size**, **Marker masks** and **Reorder**, see chapter "2D view toolbar" on page 38.

c. Camera selection toolbar for selection of displayed cameras, see chapter "Modifying the 2D view" on page 39.

d. Captured Oqus video files are displayed in the 2D view window, see chapter "2D view window" on page 36.

e. Automatic image size reduction when changing the capture rate from the **Camera settings** sidebar, see chapter "Camera settings sidebar" on page 40.

f. Modified **2D view window** menu, see chapter "2D view window menu" on page 44.

3. 3D data overlay on video from Oqus cameras, see chapter "3D data overlay on video" on page 260.
4. Undo/Redo on trajectory and bone operations, see chapter "Edit" on page 93.
5. Menu changes

   a. All of the pop-up menus in the main menus, see chapter "Menus" on page 92.

   b. More keyboard shortcuts, see chapter "Keyboard shortcuts" on page 100.

   c. More options in the **Trajectory info window** menu, see chapter "Trajectory info window menu" on page 69.

   d. Most recently used file on the **File** menu, see chapter "File" on page 92.

6. Auto backup after a measurement, see chapter "Auto backup" on page 249.
7. Updated instructions for marker masking, see chapters "How to use marker masking" on page 264 and "How to use auto marker masking" on page 265.
8. More **Advanced timing** settings, see chapter "Advanced (Timing)" on page 132.
9. Description of the synchronization options for Oqus, see chapters "Using Sync out for synchronization" on page 380, "Using Trig in for synchronization" on page 382 and "Using External timebase for synchronization to a periodic signal" on page 382.
10. Frame synchronization of USB analog boards, see chapter "Connection of analog board" on page 356 and "External sync" on page 164.
11. Changed how the **Bone length** tolerance option is used by the **6DOF calculation**, see chapter "Calculating 6DOF data" on page 290.
12. Added 6DOF body name in the **Data info** window, see chapter "6DOF data information" on page 81.
13. Residuals and Force plate locations are added in MAT files, see chapter "MAT file format" on page 323.
14. Changed real time frequency setting during a measurement, see chapter "Marker real time frequency while capturing" on page 248.
15. Updated instructions for how to connect analog board and force plates, see chapters "Connection of analog board" on page 356 and "Connection of force plate" on page 359.

16. Added **Any** setting for the trigger edge of the **External trigger**, see chapter “External trigger (advanced)” on page 133.
17. Start a capture directly with Ctrl+M, see chapter “Introduction to capture” on page 246.
18. The cameras starts in the same mode as the previous measurement or preview, see chapter “Introduction to capture” on page 246.
19. Different analog sample rates on separate boards, see chapter “Board settings” on page 162.

QTM 2.1

The following major changes have been added to the manual for QTM 2.1.

1. Improved handling of analog data
    a. Set time scale for the analog and force RT plots, see chapter “GUI” on page 218.
    b. Synchronized start in real-time/preview mode, see chapter “How real time works in QTM” on page 271.
    c. All of the analog frames sent on RT output, see chapter “How real time works in QTM” on page 271.
    d. 6 decimals in the analog TSV export, see chapter “Analog data (_a.tsv)” on page 316.
    e. Noraxon and EMG can be started with trigger button in RT to get synchronized data, see chapters “Making a measurement with Noraxon” on page F - 6 and “Making a measurement with Mega” on page F - 16.

       TIP: Activate the EMG and the synchronization when the setup is finished to minimize the number of times you press the trigger button.
2. New preview mode in Oqus called Marker intensity, which shows a color coded video image with the marker settings, see chapter “Video preview in QTM” on page 256.
3. Upload and download linearization files from Oqus cameras, see chapter “Camera linearization parameters” on page 116 and “Linearization of the cameras” on page 229.
4. New camera settings
    a. New marker masking setting including auto marker masking, see chapter “Marker masking” on page 146.
    b. New name on active markers, see chapter “Marker type” on page 145.
5. New timing settings for Oqus including custom sync out signal, see chapter “Advanced (Timing)” on page 132 and “Synchronization output” on page 137.
6. Go to individual camera settings from the **2D View window** menu, see chapter “2D view window menu” on page 44.
7. Wand calibration uses a fixed number of frames to improve the calibration result, see chapter “Calibration dialog” on page 236.
    a. Change the number of frames used in a calibration, see chapter “Maximum number of frames used as calibration input” on page 120.

8. DIFF format, see chapter “Export to DIFF format” on page 321 and “DIFF export” on page 213.
9. MAT export settings and Mat export included in processing steps and batch processing, see chapter “Matlab file export” on page 212.
10. How to connect force plates, see chapter “Connection of force plate” on page 359.
11. Tips on how to set the camera settings for Oqus, see chapter “Tips on marker settings in QTM” on page 250.
12. Description of how to synchronize external hardware, see chapter “How to synchronize external hardware” on page 379.
13. Instructions for MotionBuilder plug-in, see chapter “How to use Qualisys MotionBuilder Plugin” on page H - 1.
    a. Installing licenses for plug-ins, see chapter “Enter plug-in license” on page 7.
14. Camera tracking information in **File information**, see chapter “3D tracking test” on page 284.
15. Display different settings in the **Project options** dialog, see chapter “Change displayed pages in Project options” on page 105.
16. Modified QDS icon for Oqus disabled networks, see chapter “Advanced (network settings)” on page 339.

QTM 2.0.379

The following major changes have been added to the manual for QTM 2.0.379.

1. Individual camera settings.
    a. Oqus cameras, see chapter “Advanced (Oqus)” on page 143.
    b. Marker discrimination settings for ProReflex cameras, see chapter “Camera settings (ProReflex)” on page 156.
2. Mixed Oqus camera systems, see chapter “Mixing Oqus camera types” on page 345.
3. QDS wizard is improved to handle WLAN setup and better handling of network interface, see chapter “Oqus wireless camera setup wizard” on page 335 and “Oqus network configuration wizard” on page 333.
4. Labelling of EMG channels, see chapter “Noraxon QTM settings” on page F - 8 and “Mega QTM settings” on page F - 18.
5. Auto-close on new and open file, see chapter “GUI” on page 218.
6. Export to MAT file, see chapter “Export to MAT format” on page 322.
7. Align global coordinate system to a line, see chapter “Rotate axis to line” on page 129.

QTM 2.0.365

The following major changes have been added to the manual for QTM 2.0.365.

1. Oqus instructions and settings. The most important chapters are:
2. "Setting up the system (Oqus)" on page 329
   - "Oqus video capture" on page 256
   - "Oqus - Camera manual" on page A - 1
   - "QFI (Oqus)" on page D - 1
   - The settings are described under the chapter "Capture" on page 107.
   - Marker masking, see chapter "How to use marker masking" on page 264.
3. Real time capture, see chapter "Real time in QTM" on page 271.
4. Improved batch processing settings, see chapter "Batch processing" on page 279.
5. Improved AIM with multiple AIM models, see chapter "AIM" on page 178.
6. Quick identification method, see chapter "Manual identification of trajectories" on page 300.
7. Improved reprocessing, see chapter "Reprocessing a file" on page 281.
8. New 6DOF tracking, see chapter "6DOF tracking" on page 179 and "6DOF tracking identifies rigid bodies from the 3D data of a measurement and calculates their translation and rotation with respects to the global coordinate system. It is activated with Calculate 6DOF on the Processing page in the Project options dialog and is controlled by the 6DOF tracking parameters, see chapter "6DOF tracking" on page 179." on page 285.
   a. Changed settings for local coordinate system, see chapter "Coordinate system for rigid body data" on page 184.
   b. Post-processing 6DOF data, see chapter "Calculating 6DOF data" on page 290.
   c. Reset rotation, see chapter "Rigid bodies" on page 180.
   d. New options for translate and rotate, see chapter "Translate body" on page 182 and "Rotate body" on page 183.
   e. Track several bodies with the same definition, see chapter "Definition of 6DOF bodies" on page 287.
9. New tracker, see chapter "3D Tracking" on page 172.
10. Noraxon EMG, see chapter "Noraxon EMG" on page F - 1.
11. Mega EMG, see chapter "Mega EMG" on page F - 10.
12. USB-2533 analog board "USB-2533" on page E - 4.
13. Linearization process, see chapter "Linearize a camera" on page 229.
14. Center on the selected trajectory, see chapter "Trajectory info window menu" on page 69.
15. Import video link, see chapter "Import video link" on page 366.
16. Video analysis program Tema QVA, see chapter "TEMA QVA" on page 262.
   a. Import tracked data from Tema, see chapter "Analyzing data in video files" on page 261.
17. Kistler COP correction, see chapter "Kistler COP Correction" on page 194.

18. Force plate display without analog data, see chapter “Viewing force data” on page 311.
19. C3D export without 3D data, see chapter “Export to C3D format” on page 320.

QTM 1.10.282

The following major changes have been added to the manual for QTM 1.10.282.

1. Description of how AIM works, see chapter “How AIM identifies trajectories” on page 307.
2. The video data can be played with an offset, see chapter “Video capture” on page 365.
3. Acceleration of 6DOF data can be plotted from the Data info window, see chapter “6DOF data information” on page 81.
4. New instructions for installation of several USB A/D boards, see chapter “Installing several USB-1616FS boards” on page E - 10.
5. Some advice on how to set the 3D Tracker parameters, see chapter “Advice on how to set 3D Tracker parameters” on page 283.
6. New options on the **Timeline control** bar, see chapter “Timeline control bar” on page 61.

QTM 1.9.254

The following major changes have been added to the manual for QTM 1.9.254.

1. The coordinate system can be translated and rotated to any position, see chapter “Transformation” on page 128.
2. Forces and forces plates are shown in **3D view** windows, see chapter “Viewing force data” on page 311.
3. There are new keyboard shortcuts in the **Project options** dialog, see chapter “Project options shortcuts” on page 102.
4. In preview you can use **Viewfinder** to view a greyscale image of the camera view, see chapter “Viewfinder” on page 48.
5. QTM can track the 2D markers with 2D tracking, see chapter “2D tracking of data” on page 298.
6. Capture files can be reprocessed with any tracker (3D, 6DOF or 2D), see chapter “Reprocessing a file” on page 281.
7. The **Processing** page in the **Project options** dialog has changed, see chapter “Processing” on page 170.
8. There is a new option when exporting to TSV and C3D **Exclude non-full frames from beginning and end where any of the labeled trajectories are not found**, see for example chapter “TSV export” on page 208.
9. There are new options in the **Calibration results** dialog. Among other things it is possible to export calibrations and to load old calibrations, see chapter “Calibration results” on page 237.
10. The rotation angles in QTM can be defined to any wanted definition, see chapter “Defining Euler angles in QTM” on page 326.

11. You can plot the velocity of 6DOF bodies, see chapter “6DOF data information” on page 81.
12. The coordinate system for rigid body data can be changed to other coordinate systems than the global coordinate system, see chapter “Coordinate system for rigid body data” on page 184.
13. **Acquire body** is improved so that you do not need to be 3D tracking when using it, see chapter “Acquire body” on page 185.
14. There is a new option in **Rotate body**, see chapter “Rotate body” on page 183.
15. The description of how to use 6DOF analog output is improved, see chapter “6DOF analog output” on page 294.
16. Description of how to design a 6DOF body, see chapter “How to design a 6DOF body” on page 286.
17. Force plate control settings for Kistler force plates have been corrected, see chapter “Force plate control settings” on page 167.
18. There is an option to limit the maximum refresh rate of visualization in RT output, see chapter “RT output” on page 203.
19. Detailed descriptions of Fixed camera calibration and **Validation** used for Fixed camera calibration have been added to the manuals **Fixed camera system - Installation procedure** and **QTM - Marine manual**. Contact Qualisys AB about the two manuals.
20. Better descriptions of how to use AIM, see chapter “Generating an AIM model” on page 302.
21. More troubleshooting, see respective chapters under “Troubleshooting QTM” on page 387.

Glossary

2D
Two dimensions

2D marker ray
The 2D position of a marker projected into the 3D space by using the position of the camera.

2D view window
Window with the 2D views of the cameras.

3D
Three dimensions

3D point
Point that is specified with the three coordinates of the 3D space.

3D tracking
Tracker that uses the 2D data of all cameras in the system to calculate marker positions in three dimensions.

3D view window
Window with a 3D view calculated from the 2D data.

AIM (Automatic Identification of Markers)
Process that automatically identifies the trajectories as being part of a defined trajectory.

Aperture
The size of the opening in the camera's lens. This opening can be adjusted with the adjustment ring.

Bit
Computer unit, which can be either 0 or 1.

BNC
Type of contact for coaxial cables.

Bone
Visible connection between two trajectories in the 3D view.

Byte
Computer unit. 1 byte = 8 bit

C3D
Standard file format in motion capture

Calibration
Process that defines the position of the cameras in the 3D space. The calibration is used for the 3D reconstruction.

Calibration kit
Equipment that is needed for a wand calibration, e.g. calibration wand and L-shaped reference structure.

Capture
Measurement which collects several frames at a fixed frame rate.

Capture file
A qtm-file with motion capture data (.qtm).

Capture rate
Frame rate in Hz that is used for the motion capture.

Capture view
View that is used during motion capture.

Coordinate system
A system of axes which describes the position of a point. In QTM all of the 3D coordinate systems are orthogonal, right hand systems.

Coordinate system of the motion capture
The coordinate system which is defined with the calibration process.

Data info window
Window with data for the current frame.

Discarded trajectories window
Window with deleted trajectories.

Extended calibration
Method that extends the motion capture volume, when using Wand calibration.

External trigger
Device that triggers the motion capture system to start the measurement.

Field of view (FOV)
The MCU's view, vertical and horizontal on a specific distance from the camera.

File view (File mode)
View that is used when a motion capture is finished and for saved files.

Fill level
The percentage of the capture period in which a trajectory has 3D data (been visible).

Focus
Changes the focal length of the camera to achieve a clear image.

Frame
Single exposure of the camera system.

Frame rate
Frequency of the motion capture.

Gap fill
Function that calculates a probable path between trajectory parts to associate them.

Global coordinate system
In QTM it is the same as the coordinate system of the motion capture, which is defined by the calibration.

Immediate mode
Mode where a camera continuously captures data and sends it to the measurement computer. The data is sent as soon as it has been captured.

IR
Infrared

IR marker
A marker which reflects or transmits IR light.

Label
Name of a trajectory in the Identification windows.

Labeled trajectories window
Window with identified trajectories.

LED
Light Emitting Diodes

Linearization
Correction data which is needed for each camera to make the capture as good as possible.

Marker
Item that is attached to the moving object to measure its position.

Marker – Active
Marker with an infrared LED that is activated by the camera's flash in each frame.

Marker – Passive
Marker with reflective material.

Marker (3D view)
Sphere that represents a trajectory in 3D views.

Marker discrimination
Option that reduces non-wanted reflections or marker sizes during capture.

Markers mode
The default mode in 2D view windows, which shows the markers captured by the camera.

Max residual
Maximum distance for a 2D ray to be included in a 3D point during tracking.

Measurement computer
Computer which is connected to a camera system, which must have the QTM application installed.

Measurement range
The range that is set with the boxes on the Timeline control bar. It defines the frames which is included in the analysis.

Motion capture
Measurement which records a motion.

Open GL
Standard graphic presentation language.

Plot window
Window with data plots.

Pretrigger
Setting in QTM where the frames before the trigger are saved in the MCU's memory and are then added to the measurement when a trigger event occurs.

Preview mode
Mode when QTM is showing the measured data before the start of a capture.

QFI
Program for installing firmware in the MCU.

Real-time output (RT output)
Function which exports 3D, 6DOF and Analog data in real-time to a remote computer.

Remote computer
Computer which receives 6DOF data from the RT output.

Residual
In most cases in QTM this is the minimum distance between a 2D marker ray and its corresponding 3D point or an average of this measure.

Residual (3D)
The average of the different residuals of the 2D marker rays that belongs to the same 3D point.

Residual (calibration)
The Average residual in the Calibration results dialog is the average of the 3D residuals of all the points measured by the camera during the calibration.

TCP/IP
Protocol for communication between computers.

Trace
Traces the position of a trajectory in the 3D view.

Trace range
Range of frames for which the trace is shown.

Tracking
Process that calculates 3D data or 6DOF data.

Trajectory
3D data of a marker in a series of frames.

Trajectory info windows
Windows with trajectories and 3D data.

Trigger
Signal that starts a measurement.

Tripod
A very stable three-legged stand.

TSV (Tab Separated Values)
File format where the data is separated with the TAB character.

Unidentified trajectories window
Window in QTM with unidentified trajectories.

USB
Hardware interface for connecting peripheral units to a computer.

Wand
T-shaped object that is used in Wand calibration to scale and define the axes of the coordinate system of the motion capture.

Wand calibration
Calibration method which uses a wand and an L-shaped structure to calibrate.

View window
Window in QTM which shows 2D, 3D or Video views.

WLAN
Wireless local area network.

Volume
The defined measurement's height, length, depth.

Zoom
Zoom in to get a close-up of your 3D view or zoom out to see more of the page at a reduced size.

Index

D

M

N

O