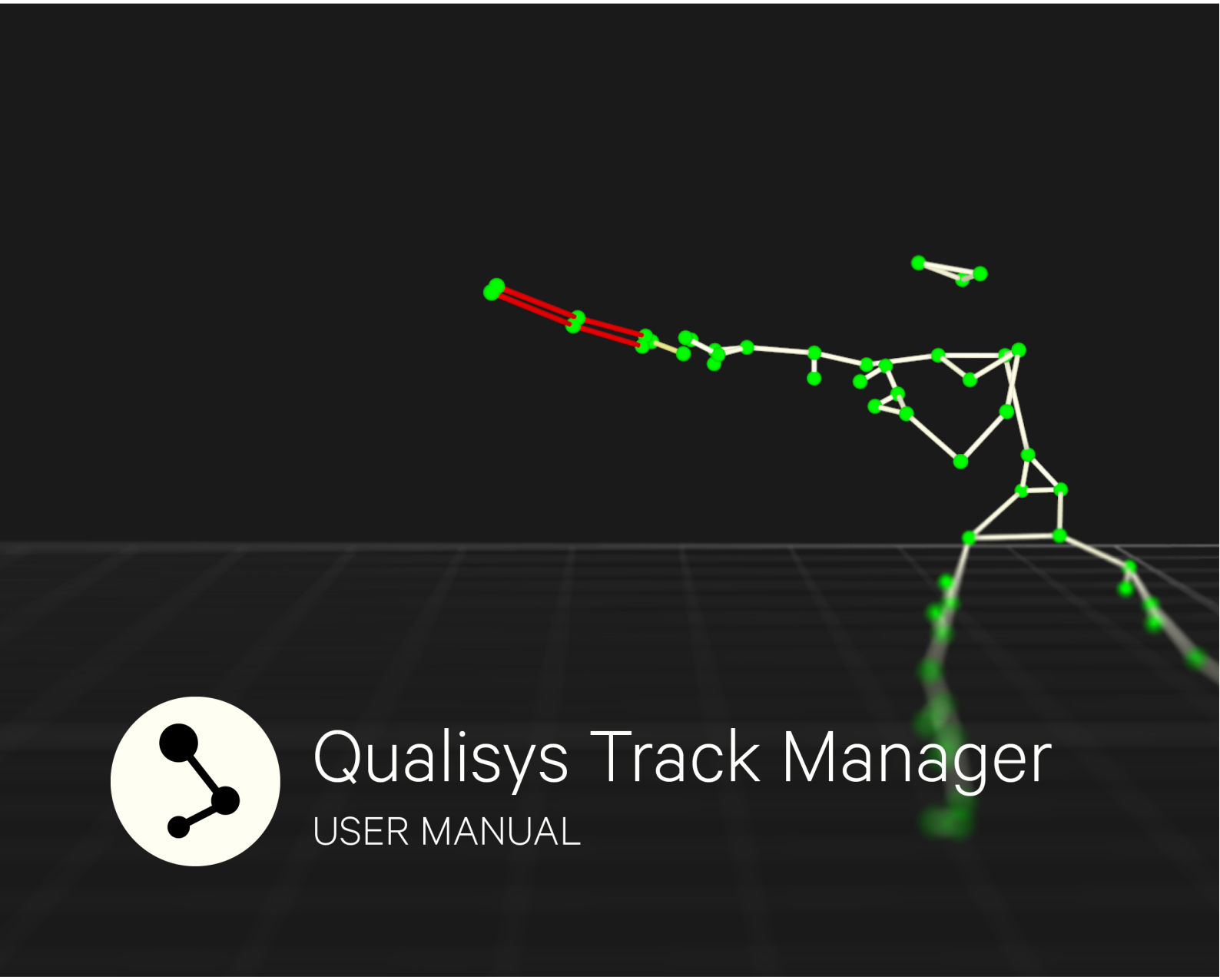


Qualisys Track Manager

USER MANUAL

QUALISYS AB

Kvarnbergsgatan 2 • 411 05 • Gothenburg • Sweden
Phone: +46 (0)31 336 94 00 • Fax: +46 (0)31 336 94 20
sales@qualisys.com • support@qualisys.com

Qualisys Track Manager

Version: 2020.2

Qualisys AB, 6/26/2020

www.qualisys.com

No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language in any form by any means without the written permission of Qualisys AB.

In no event shall Qualisys AB be liable for any incidental, indirect, or consequential damages whatsoever (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or any other pecuniary loss) arising out of the use of or inability to use the software or hardware.

Copyright © 2020 Qualisys AB ® All Rights Reserved.

Qualisys Track Manager and QTM are registered ® trademarks of Qualisys AB.

Table of Contents

Getting Started **1**
Qualisys Track Manager 1
Introduction 1
Organization of the manual 1
IR radiation notice 3
Safety notice 3
System setup 4
Hardware requirements 4
Plugin requirements 6
Hardware installation 6
Software installation 7
QTM registration 7
Enter plug-in license 7
Windows user permissions 9
Hardware configuration 10
Training resources 11
Store real-time data 11
QTM user interface **13**
QTM specific user interface 13
RT(Preview)/File mode 13
Window types in QTM 13
Projects 14
Using projects 14
Projects with qualified users 14
Projects with students 15
Using projects on multiple computers 15
QTM startup procedure 15
Creating a new project 17
Manage projects 18

Project view 19
Project data tree 19
Details 21
Backup of project settings 22
Project presets 22
Project folder 23
QTM windows 25
Main window 25
Main status bar 26
View windows 27
2D view window 27
2D view toolbar 29
Modifying the 2D view 30
Camera settings sidebar 31
External video devices in 2D view 36
2D view window menu 39
3D data overlay 41
Review camera settings 41
3D view window 43
Using and modifying the 3D view 44
Change the viewpoint of the 3D view 44
3D view toolbar 46
Trajectories in 3D views 48
Bones in 3D views 49
6DOF bodies in 3D views 50
Skeletons in 3D views 51
Rays in 3D views 52
Camera view cones in 3D views 53
Volumes in 3D views 54
Other objects in 3D views 56
3D view window menu 57
Bone menu 58
Timeline control bar 59
Timeline parameters 60
Events menu 61
Trajectory info windows 63
Data in Trajectory info windows 63
Sort trajectories 65

Select and move trajectories 65
Overlapping trajectory parts 66
Trajectory info window menu 68
Plot 3D data 72
Split part after current frame 72
Delete trajectories 72
Swap parts 73
Analyze 74
Filters 74
Calculation 75
Output name 77
Label lists 77
Labels menu 77
Trajectory Editor window 79
Plot area 79
Trajectory Editor toolbar 80
Points of Interest sidebar 82
Settings sidebar 83
Trajectory Editor window menu 84
Data info window 85
Data info window menu 85
Data types 86
2D data information 86
6DOF data information 87
Skeleton data information 89
Analog data information 91
Force data information 92
Calculate 93
Messages window 94
Plot window 94
Toolbar 95
Legend box 96
Window layout 98
Windows dialog 99
Menus 101
File 101
C3D import 102
Edit 102

View 103
File Information 105
Play 105
Capture 106
AIM 106
Skeleton 107
Tools 107
Window 107
Help 108
Popup window menus 108
Toolbars 109
Standard toolbar 109
Playback toolbar 109
Capture toolbar 109
GUI Control toolbar 110
AIM toolbar 110
Trajectory toolbar 110
Keyboard shortcuts 112
Menu shortcuts 112
2D/3D view shortcuts 113
Trajectory info window shortcuts 114
Playback keys 115
Project options shortcuts 116
Trajectory Editor shortcuts 118
Dialogs 119
QTM dialogs 119
Project options dialog 119
Project options 121
Input Devices 121
Camera system 123
Marker capture frequency 123
Real time frequency 123
Camera system settings 124
Locate system 124
Finding camera system 124
Auto order (Miqus) 125
Camera system information 125
Cameras 127

Camera settings 128
Camera Mode 128
Marker settings 128
Capture rate 128
Exposure time 128
Marker threshold 129
Marker type 129
Image size 130
Marker masking 131
Marker circularity filtering (Oqus) 132
Marker limits 132
Smallest 133
Largest 133
Max number of markers per frame 133
Exposure delay 133
Sensor mode 134
Video settings 134
Capture rate 134
Exposure time 134
Flash time 135
Gain 135
Image size 135
Image resolution 136
Video compression 137
FFDS-MJPEG codec settings 138
Color/Grayscale 138
Auto exposure 138
Color temperature 139
Sensor mode 139
Active filtering (Oqus) 139
Lens aperture 140
Lens focus distance 140
Linearization 141
Camera linearization parameters 141
Calibration 144
Calibration type 144
Wand calibration 144
Calibration kit 144

Coordinate system orientation and translation 145
Maximum number of frames used as calibration input 146
Apply coordinate transformation 146
Fixed camera calibration 147
Reference marker locations 147
Camera locations and markers seen by each camera in order from left to right 147
Apply coordinate transformation 148
Transformation 149
Rotate axis to line 150
Current calibration 152
Calibration quality 153
Synchronization 154
Wireless trigger 155
Trigger ports 156
Event port (Camera Sync Unit) 157
Pretrigger (Oqus) 158
External timebase 159
Timestamp 162
Synchronization output 162
Measurement time (Camera Sync Unit) 165
Analog boards 166
Analog board settings 166
Sample rate 167
Board settings 168
Force plate control 168
Compensate for analog offset and drift 168
Channels 170
Force plate control settings 173
Analog board type 173
Force plate auto-zero (Kistler) 174
Force plate control list 174
AV devices 176
AV device settings 177
Force plates 178
AMTI Gen5 178
EMGs 180
Processing 181

2D Preprocessing and filtering 182
Non-circular marker settings (Oqus) 183
Filtering 183
3D Tracking 183
3D Tracker parameters 184
Prediction error 184
Maximum residual 185
Minimum trajectory length 185
Minimum ray count per marker 185
Ray length limits 186
Rays 186
Auto join 186
Bounding box restricting 3D data 186
Auto range 187
2D tracking 187
Tracking settings 187
Auto join 188
2D to 3D settings 188
Twin System 189
Twin Slave System 189
Capture Frequency 190
Trajectory Conflicts 190
Twin System Calibration 191
Twin System Calibration dialog 192
Trajectories 192
Gap Fill Settings 193
AIM 193
AIM models 194
AIM model application parameters 194
Skeleton solver 195
6DOF tracking 196
6DOF Tracker parameters 196
Rigid bodies 196
Translate body 199
Rotate body 200
Coordinate system for rigid body data 202
Acquire body 203
Smoothing 6DOF data 204

Rigid body Mesh Settings dialog 205
Compatibility of meshes 206
Euler angles 206
Select Euler angle definition 207
Definition of rotation axes 208
Force data 210
General settings 210
Force plates 210
Force plate 211
Force plate type 211
AMTI force plate calibration parameters 211
Force plate dimensions 212
Inverted sensitivity matrix 212
AMTI force plate settings 212
Kistler force plate calibration parameters 213
Force plate dimensions 214
Kistler COP Correction 214
Kistler scaling factors 214
Calculating scaling factors with external amplifier 215
Maximum measurable force 215
Kistler force plate settings 216
Bertec force plate calibration parameters 216
Dimensions 217
Calibration matrix 217
Bertec force plate settings 217
AMTI 8-Channel force plate calibration parameters 218
Force plate dimensions 219
Calibration matrix 219
AMTI 8 Ch force plate settings 219
Generic 6 ch (c3d type-2) force plate calibration parameters 220
Force plate dimensions 220
Generic 6 ch (c3d type-2) force plate settings 220
Generic 8 ch (c3d type-3) force plate calibration parameters 221
Force plate dimensions 221
COP correction 221
Generic 8 ch (c3d type-3) force plate settings 221
Generic 6 ch with matrix (c3d type-4) force plate calibration 222
Force plate dimensions 223

Calibration matrix 223
Generic 6 ch with matrix (c3d type-4) force plate settings 223
Force plate location 224
COP (Center Of Pressure) threshold 227
Force plate settings status window 227
Real-Time output 227
6DOF analog export 228
Analog channel settings 229
Test output 230
Range calibration 231
TSV export 231
Data type to export 231
General export settings 232
Settings for 3D data export 232
C3D export 233
Settings for 3D data export 233
Select label output format 233
Zero baseline 234
Select event output format 234
Exported units 234
Matlab file export 234
AVI Export 235
Window settings 235
Video settings 236
FBX export 238
Start program 239
GUI 241
2D view settings 243
3D view settings 244
Static mesh objects 247
Static Mesh Settings dialog 248
Miscellaneous 250
Folder options 250
Startup 251
Events 252
Event shortcuts 252
Making a measurement 253
Connection of the camera system 253

Camera system connection 253
Outline of how to locate the camera system 253
Waiting for cameras to synchronize after new file 254
Optimizing the camera system and settings 254
Camera positioning 254
2D motion capture 255
3D motion capture 255
Identifying and ordering the cameras in QTM 255
Identifying the cameras 256
Automatic ordering of the cameras (Miqus) 256
Tips on setting aperture and focus 257
Tips on marker settings in QTM 258
Delayed exposure to reduce reflections from other cameras 259
IR markers 261
Valid marker types 261
Passive vs Active markers 261
Standard marker sizes 261
Marker size 262
Marker placement 262
Marker maintenance 262
Linearization of the cameras 263
Camera linearization 263
Linearize a camera 263
Concept of QCC linearization 263
Linearization instructions 264
Linearization file warning 267
Calibration of the camera system 269
Introduction to calibration 269
Outline of how to calibrate (Wand calibration) 269
Calibration dialog 270
Calibration quality 270
Calibration timing 271
Sound 271
Linearization parameters 271
Calibration results 271
Quality results 272
View cal. 273
Calibration failed 273

Calibration quality warning 273
Wand calibration method 275
Calibration tips 275
How to move the wand 275
Extended calibration 276
Refine calibration 277
Carbon fiber wand kit (600 mm) 278
Translating origin to the floor 278
Fixed camera calibration method 279
Reprocess calibration 279
Capturing of measurements 281
Introduction to capture 281
Outline of how to capture 281
Start capture 282
Capture period 283
Capture delay and notification 283
Automatic capture control 283
Camera systems settings 283
Batch capture 283
Auto backup 284
Store real-time data 284
Qualisys video capture 286
Using Qualisys video cameras 286
Calibrating Qualisys video cameras 287
Capture streaming video 287
Miqus Video camera 287
Oqus color video camera 287
Streaming Video settings 288
Maximum capture rate for streaming video 288
Miqus Video 288
Oqus 2c 289
Capture video with in-camera MJPEG 289
Capture high-speed video 289
Information about high-speed video capture 290
Outline of how to capture high-speed video 290
Codecs for Oqus high-speed video files 291
Video preview in QTM 292
Qualisys video files 294

3D data overlay on video 295
Active filtering for capturing outdoors 296
How to use active filtering 296
Marker masking 297
How to use marker masking 297
How to use auto marker masking 298
Marker circularity filtering (Oqus) 299
How to use non-circular marker settings 299
Twin systems 301
How to use frame synchronized twin systems with separate volumes 301
How to use start synchronized twin systems with separate volumes 304
Performing a Twin calibration 307
Working with QTM twin files 309
Merging Twin files in reprocessing 310
Twin 3D data interpolation 310
Twin system with a shared volume 311
Real time mode 313
Real time in QTM 313
How real time works in QTM 313
Real time latency 315
Outline of how to use real time 315
Server mode 317
Mobile apps 317
Master/Slave mode in mobile apps 317
Visual3D integration 318
OSC integration 319
LabVIEW, Matlab and Simulink integrations 319
VRPN integration 320
Vizard example script 320
Processing measurement data 325
Load project settings and calibration 325
Opening a project or restoring project settings 325
Loading a saved calibration 325
Data processing steps 326
Introduction to data processing 326
Reprocessing a file 327
Changing calibration 328
Batch processing 329

Processing 2D data 330
How to use circularity filter 330
How to use software marker masks 331
How to use marker size filter 332
Tracking the measurements 334
Introduction to tracking 334
3D tracking measurements 334
Advice on how to set 3D Tracker parameters 334
3D tracking test 335
Tracking and accuracy 336
6DOF tracking of rigid bodies 336
6DOF versus 3D 337
Creating 6DOF bodies 337
How to design a 6DOF body 337
Definition of 6DOF bodies 338
Define 6DOF bodies using Acquire body method 339
Definition of local coordinate system 341
Tracking 6DOF bodies 341
Calculating 6DOF data 342
Virtual markers calculated from 6DOF data 343
Rotation angles in QTM 344
6DOF real-time output 346
6DOF analog output 346
Analog output hardware 346
Rigid body meshes 347
Sharing a file with a rigid body mesh 348
Examples of how to use 6DOF bodies 348
How to use 6DOF bodies 348
How to use 6DOF bodies in an AIM model 348
How to use virtual markers in an AIM model 349
How to improve 6DOF data with marker filtering (Oqus only) 350
Tracking of skeletons 350
Marker sets for skeleton tracking 351
Qualisys Animation Marker Set 351
Qualisys Sports Marker Set 352
Adding extra markers to a skeleton 353
Skeleton marker label mapping 353
Automatic labeling of markers for skeleton tracking 354

Generating an AIM model for animation 354
Using AIM for sports and biomechanics 355
Skeleton calibration 356
T-pose 357
Partial skeleton calibration 357
How to modify the skeleton definition 358
Scale factor 359
Skeleton marker weights 359
How to measure skeleton data 360
Export and streaming of skeleton data 360
2D tracking of data 360
Using 2D tracking 361
Identification of trajectories 362
Manual identification of trajectories 362
Tips and tricks for manual identification 363
Automatic Identification of Markers (AIM) 364
Generating an AIM model 364
Guidelines for data added to AIM models 369
How to verify and edit AIM bones 370
Applying an AIM model 372
How AIM identifies trajectories 374
AIM models for multiple subjects in the same measurement 375
Unique subjects 375
Similar subjects 376
Use generic AIM model to create unique models 376
Editing of trajectories 377
Gaps 377
Identification and selection of gaps 378
Filling of gaps 378
Spikes 380
Detection and selection of spikes 381
Smoothing 381
Adding virtual trajectories 382
How to use events 383
Adding events 383
Viewing and editing events 384
Exporting events 385
Force data calculation 386

Calculating force data 386
Viewing force data 386
Data export to other applications 389
Export to TSV format 389
TSV file formats 390
Motion data (.tsv) 390
6DOF data format 392
Skeleton data (_s_name.tsv) 393
Analog data (_a.tsv) 394
Force data (_f.tsv) 396
Eye tracker data (_g.tsv, _e.tsv) 397
Export to C3D format 397
C3D file format 398
Export to MAT format 399
MAT file format 399
Export to AVI file 404
Export to FBX file 406
Euler angles 407
Defining Euler angles in QTM 407
System hardware 409
Qualisys cameras and devices 409
Qualisys camera types 409
Marker cameras 409
Video cameras 410
Streaming video 411
Miqus Video 411
Oqus color video camera (2c-series) 411
Miqus Hybrid 411
Oqus high-speed video camera 412
Weather protected cameras 412
Underwater systems 412
MRI compatible cameras 413
Description of Qualisys devices 414
Description of Arqus cameras 414
Arqus camera: front side 414
Arqus camera: back side 415
Description of Miqus cameras 416
Miqus camera: front side 416

Miqus camera: back side 417
Description of Camera Sync Unit 418
Camera Sync Unit: front side 418
Camera Sync Unit: back side 419
Description of Oqus devices 420
Oqus camera display 420
Oqus camera connectors 421
Oqus Sync Unit 423
Setting up the system 424
Camera system setup: technical notes 424
Mixing Arqus and Miqus 425
Adding Miqus Hybrid or Miqus Video cameras 425
Setting up a Qualisys system with an Ethernet switch 425
Setting up an Arqus system 426
The Arqus startup sequence 427
Setting aperture and focus 428
Setting up a Miqus system 428
The Miqus startup sequence 430
Setting aperture and focus 430
Setting up Miqus Hybrid or Video for markerless mocap 431
Setting up an Oqus system 432
Oqus startup sequence 433
Setting the aperture and focus 434
Setup Oqus system for wireless communication 435
Setting up a mixed system 435
Network configuration 437
Network card setup 437
QDS 437
QDS menu 437
Network configuration wizard 439
Advanced (network settings) 441
QDS conflict 442
Firmware update 443
Firmware update when locating system 443
Firmware update when opening a new file 443
Short range active marker 445
Short range active marker hardware 445
Short range driver 445

Active markers and IR eye 447
Battery and charging 447
QTM settings for the active marker 447
How to use sequential coding 448
Passive and Active mode 448
When to use short range active marker 449
Short range active marker specifications 449
How to use analog boards 450
Connection of analog board 450
Analog offset warning 451
Analog saturation warning 452
Analog devices 452
How to use force plates 454
Connection of force plate 454
Connecting Kistler force plates 454
Connecting Kistler 5695A DAQ 457
Connecting AMTI and Bertec force plates 458
Connecting AMTI Gen5/OPT-SC/AccuGait Optimized 460
Supported force plate devices 462
How to use EMG 463
EMG devices 463
How to use video devices 464
Video capture 464
Video capture with Blackmagic Design cards 464
Installing the BlackMagic Design card 464
Connecting a video source to the Intensity Pro card 465
Connecting a video source to the Decklink Mini Recorder card 465
Using Blackmagic Design video source in QTM 466
Settings on Sony HDR-CX330 468
Settings on Sony HDR-CX430V 470
Panasonic AW-HE2 472
DV/webcam devices 473
Compression of video from video cameras 473
Video offset 473
Selecting audio source 474
Import video link 474
Timing hardware 475
How to use external trigger 475

How to use pretrigger 475
Measurement with analog capture while using pretrigger (Oqus) 475
How to use external timebase 476
External timebase connection 476
Using External timebase for synchronization to a periodic TTL signal 476
Using External timebase for synchronization to a time code signal 478
Using Qualisys video with External timebase 479
External timebase with bursts of signals with constant period (cycle) time and with delays between the bursts 479
How to use PTP sync with an external clock master (Camera Sync Unit) 480
How to use PTP sync with an external clock master (Oqus) 482
Synchronizing external hardware 484
How to synchronize external hardware 484
Using Sync out for synchronization 484
Using Trig in for synchronization 486
Using Oqus sync unit for synchronization 486
Using SMPTE for synchronization with audio recordings 488
Troubleshooting QTM 491
Troubleshooting connection 491
Troubleshooting calibration 493
Troubleshooting capture 494
Troubleshooting tracking 497
Troubleshooting reflections 499
Troubleshooting force calculation 500
Troubleshooting 6DOF 501
Troubleshooting update 502
Troubleshooting Oqus wireless 503
Troubleshooting other 505
Appendix A: Qualisys camera manual ..... A - 1
Overview of camera models and sensor specifications ..... A - 1
Qualisys camera sensor specifications (marker mode) ..... A - 1
Oqus high-speed video sensor and memory specifications ..... A - 2
Qualisys video sensor specifications (in-camera MJPEG) ..... A - 3
Oqus technical reference ..... A - 4
Configurations ..... A - 4
Sensor ..... A - 4
High-speed video ..... A - 6

Optics ..... A - 6
Communication ..... A - 7
Memory ..... A - 7
Strobe light ..... A - 7
Environmental protection ..... A - 7
Mounting ..... A - 8
Oqus technical specifications ..... A - 9
Technical specifications ..... A - 9
Mechanics ..... A - 12
Physical specifications ..... A - 12
Optics and strobe ..... A - 13
How to adjust aperture and focus ..... A - 13
How to change strobe unit ..... A - 13
How to change lens ..... A - 13
TCP/IP ..... A - 14
Digital and analog I/O ..... A - 14
Trigger input ..... A - 14
Synchronization input ..... A - 14
Synchronization output ..... A - 15
Control connections ..... A - 15
Definition of Oqus types ..... A - 15
Appendix B: Qualisys Firmware Installer (QFI) ..... B - 1
How to use Qualisys Firmware Installer (QFI) ..... B - 1
Appendix C: Analog input/output hardware ..... C - 1
Analog I/O hardware supported by QTM ..... C - 1
Old analog hardware ..... C - 1
Installing drivers for the A/D board ..... C - 2
USB A/D board ..... C - 2
USB-2533 ..... C - 2
Installing the USB-2533 board ..... C - 3
USB-1608G ..... C - 5
USB-1608G specifications ..... C - 6
Installing the USB-1608G board ..... C - 6
Hardware synchronization ..... C - 8
Appendix D: Wireless EMG systems ..... D - 1
EMG systems in QTM ..... D - 1
Noraxon EMG ..... D - 1
Noraxon installation ..... D - 1

Noraxon computer connection ..... D - 2
Noraxon trigger connection ..... D - 2
Trigger connection to Ultium ..... D - 2
Trigger connection to DTS Desk Receiver ..... D - 2
Trigger connection to TeleMyo DTS Belt Receiver ..... D - 3
Trigger connection to TeleMyo 2400 G2 PC Interface ..... D - 4
Trigger connection to TeleMyo 2400R G2 Mini Receiver or TeleMyo 2400R G2 Analog Output Receiver ..... D - 5
Making a measurement with Noraxon ..... D - 7
Export Noraxon EMG data ..... D - 8
Noraxon QTM settings ..... D - 9
Hardware setup ..... D - 10
Hardware setup - Ultium ..... D - 11
Hardware setup - DTS Desk Receiver ..... D - 12
Hardware setup - DTS Belt Receiver ..... D - 15
Hardware setup - TeleMyo 2400T G2 ..... D - 16
Mega EMG ..... D - 17
Mega installation ..... D - 18
Mega WLAN configuration ..... D - 18
Connecting to Mega in QTM ..... D - 20
Mega hardware connections ..... D - 21
Making a measurement with Mega ..... D - 22
Mega measurement unit not responding ..... D - 23
Export Mega EMG data ..... D - 24
Mega QTM settings ..... D - 24
Delsys Trigno EMG ..... D - 26
Trigno installation ..... D - 26
Trigno computer installation ..... D - 26
Trigno synchronization connections ..... D - 28
Trigno trigger connection ..... D - 28
Trigno Measurement Time connection ..... D - 30
Making a measurement with Trigno ..... D - 31
Export Trigno EMG and auxiliary data ..... D - 32
Delsys Trigno QTM settings ..... D - 32
Cometa EMG ..... D - 33
Cometa installation ..... D - 34
Cometa synchronization setup ..... D - 35
Cometa QTM settings ..... D - 35

Making a measurement with Cometa ..... D - 36
Export Cometa EMG and IMU data ..... D - 37
Appendix E: Eye tracking hardware ..... E - 1
Eye tracking hardware in QTM ..... E - 1
Dikablis eye tracker ..... E - 2
Dikablis installation ..... E - 2
Add Dikablis eye tracker to a project ..... E - 2
Setting up eye tracking in D-lab ..... E - 4
Dikablis gaze vector calibration ..... E - 6
Making a measurement with Dikablis ..... E - 8
Process and export Dikablis gaze vector data ..... E - 9
Dikablis data in QTM ..... E - 12
Dikablis QTM settings ..... E - 13
SMI Eye Tracker Glasses ..... E - 15
SMI eye tracker installation ..... E - 15
Add SMI eye tracker to a project ..... E - 16
Making a measurement with SMI ..... E - 18
Process and export SMI gaze vector data ..... E - 18
SMI data in QTM ..... E - 20
SMI QTM settings ..... E - 21
SMI ..... E - 22
Gaze Vector ..... E - 23
Tobii eye tracker ..... E - 24
Add Tobii eye tracker to a project ..... E - 24
Setting up Tobii hardware synchronization ..... E - 26
Refinement of Tobii rigid body definition ..... E - 27
Making a measurement with Tobii ..... E - 28
Process and export Tobii gaze vector data ..... E - 28
Tobii data in QTM ..... E - 31
Gaze vector settings ..... E - 33
ASL EyeHead integration ..... E - 34
Appendix F: Rotation angle calculations in QTM ..... F - 1
6DOF tracking output ..... F - 1
Calculation of rotation angles from the rotation matrix (Qualisys standard) ..... F - 1
Calculation of other rotation matrixes ..... F - 3
Appendix G: General PAF information ..... G - 1
Project Automation Framework (PAF) ..... G - 1
PAF module installation ..... G - 1

Downloading installation files ..... G - 1
New installation ..... G - 1
Upgrading an existing installation ..... G - 4
PAF Project view ..... G - 6
Appendix H: EU customer information ..... H - 1
Waste Electrical and Electronic Equipment (WEEE) ..... H - 1
Qualisys WEEE Collection Process ..... H - 1
Appendix I: China ROHS ..... I - 1
有害物质声明 ..... I - 1
Appendix J: Open source software compliance ..... J - 1
Miqus ..... J - 1
Licenses ..... J - 1
GNU GENERAL PUBLIC LICENSE ..... J - 1
MIT License ..... J - 7
zlib ..... J - 7
Revision history I
Glossary i
Index i

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 3D, 2D and 6DOF data.

During the capture, real time 3D, 2D and 6DOF information is displayed allowing instant confirmation of accurate data acquisition. The individual 2D camera data is quickly processed and converted into 3D or 6DOF 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 eight main parts. The appendices contain detailed technical information about the cameras, instructions on how to connect external hardware, and other additional information.

Getting Started
How to get started with the QTM software and requirements for the system setup. For an overview of training resources, see the chapter “Training resources” on page 11.

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.

Troubleshooting QTM
Troubleshooting suggestions for QTM.

Appendix A
Camera manual for Qualisys cameras.

Appendix B
Manual for the Qualisys firmware installer software (QFI).

Appendix C
Installation of analog capture hardware supported by QTM.

Appendix D
Manuals for Wireless EMG systems (Noraxon, Delsys and Mega).

Appendix E
Manuals for Eye tracking systems (Dikablis, SMI and Tobii).

Appendix F
Calculation of rotation angles from the rotation matrix.

Appendix G
Information about Project Automation Framework (PAF).

Appendix H
EU customer information about the WEEE directive.

Appendix I
China ROHS information.

Appendix J
Open source software compliance.

Revision history
Information about the new features in a QTM release and new parts in the manual.

IR radiation notice

Safety notice

The Qualisys 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 Qualisys cameras belong to the exempt group according to IEC / SS-EN 62471 and are within the limits of 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 Qualisys camera system.
2. A calibration kit.
3. A stationary or portable measurement computer.
   a. For external graphic boards (Nvidia or ATI) it is required to have at least 512 MB built-in memory and support OpenGL 2.1 or higher. For integrated Intel graphics the version must be HD Graphics 4600 or later.
      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 required to have Dual/Multi Core processor of at least 2 GHz each.
   c. It is recommended to have at least 8 GB of internal memory and it is required to have 2 GB for linearization of cameras.
   d. It is required to have at least 900 pixels in vertical resolution to make sure that all of the camera settings are visible in the 2D view window.
   e. There must be an Ethernet card that supports Ethernet 802.3.
   f. For the built-in help to work you need a web browser that can handle basic HTML5, for Internet Explorer it means version 8 or later.

The following additional equipment can be used with Qualisys systems. For Arqus or Miqus systems a **Camera Sync Unit** is required for synchronizing or triggering external equipment. Alternatively, for Oqus systems without a Camera Sync Unit a trigger/sync splitter cable or an Oqus Sync Unit connected to the control port of one of the cameras can be used.

- Analog interfaces (Measurement Computing), 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, Delsys, Cometa and Mega).
- Blackmagic Intensity Pro and Decklink Mini Recorder cards for video capture, DirectShow compatible DV cameras or standard USB Web camera, with which QTM can record video sequences for documentation.
- MOTU sound device for synchronized sound recording.
- Eye tracking glasses (Tobii, SMI and Dikablis)

QTM is compatible with Windows 10 and Windows 8.1/8 (64-bit and 32-bit). It is recommended to run QTM on Windows 10. For specific hardware compatibility and requirements, see the Tables below.

Cameras		
	Win 10	**Win 8.1/8**
Arqus	Yes	Yes
Miqus	Yes	Yes
Oqus	Yes	Yes

Analog boards		
	Win 10	**Win 8.1/8**
USB-2533[1]	Yes	Yes
USB-1608G[1]	Yes	Yes
PCI-DAC6703[1, 2]	Yes	Yes
Kistler 5695B DAQ[1, 3]	Yes	Yes
AMTI Gen5[4]	Yes	Yes

1. With Instacal 6.71 or later.
2. Analog output only.
3. Requires QTM 2.6 or later. Kistler 5695A DAQ is also supported.
   Note that Kistler 5691A is not supported by QTM, because its analog board may disconnect automatically from the computer.
4. Requires QTM 2.9 or later.

Other hardware		
	Win 10	**Win 8.1/8**
Noraxon Telemyo	Yes	Yes
Noraxon DTS[1]	Yes	Yes
Mega ME6000[2]	Yes	Yes
Cometa EMG	Yes	Yes
Delsys Trigno[3]	Yes	Yes
Blackmagic Intensity Pro[4]	Yes	Yes
Blackmagic Decklink Mini Recorder[5]	Yes	Yes
Dikablis Eye Tracker[6]	Yes	Yes
SMI Eye Tracking Glasses[7]	Yes	Yes
Tobii Pro Glasses 2	Yes	Yes

1. Requires QTM 2.6 or later.
2. Requires firmware 700059 1.5.3 or 700060 1.5.3 for the ME6000 unit.
3. Requires QTM 2.9 or later. It is recommend to download the Delsys SDK via QTM via **Project Options**>**Input Devices**>**Download device drivers** to make sure that you use a compatible version. For the current version of QTM the Delsys SDK version is 3.5.6.
4. Requires QTM 2.6 or later and a computer with a PCI Express slot and that the graphic board can handle hardware acceleration. Contact Qualisys AB to make sure that it works on your computer.

5. Requires QTM 2.6 and Blackmagic drivers 9.7.2 or later and a computer with a PCI Express slot and that the graphic board can handle hardware acceleration. Contact Qualisys AB to make sure that it works on your computer.
6. Requires a separate computer with the Dikablis software (D-lab 3.45 or later).
7. Requires a separate computer or mobile device with the SMI software iView ETG version 2.7.1 (Note: earlier or later versions are not compatible).

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. The MotionBuilder plugin is included in the QTM FX version.

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

Motion capture with Qualisys Track Manager (QTM)software requires the following hardware:

- A Qualisys camera system, see chapter “Qualisys cameras and devices” on page 409.
- A computer running QTM software with a network card configured for use with the Qualisys camera system, see chapter “Network card setup” on page 437.

There is also some optional hardware which can be used with QTM, for example:

- An analog board for analog data from different analog units, such as force plate and EMG sensors, see “Analog input/output hardware” on page C - 1.
- A wireless EMG device, see “Wireless EMG systems” on page D - 1.

- An external trigger to start the measurement, see chapter “How to use external trigger” on page 475.
- A web camera, see chapter “Video capture” on page 464.

Software installation

Make sure you are logged in with an administrator account before you start installing QTM. To install the software, insert the USB installation stick. If the installation program does not auto-start, locate and execute the setup.exe file on the installation USB. You can also download the QTM installer via your registered client account at http://www.qualisys.com/my/.

Follow the instructions given during the installation. In the installer you can select the languages for the menus and dialogs in QTM. There are three available languages: English (default), Chinese and Japanese.

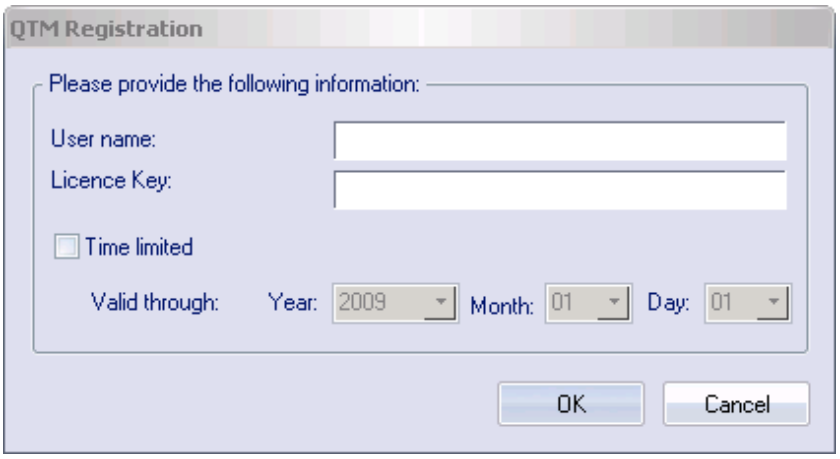
During the installation you can select the components that you want to include. The following components can be selected: Noraxon EMG driver, Instacal (A/D board driver), Kistler DAQ driver (including Instacal), FFDshow MJPEG Codec and Mega EMG driver.

Enter the user name and the license id, that you have received from Qualisys AB, 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. You can also find the latest software updates by logging in with your registered client account at http://www.qualisys.com/my/.

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 USB.

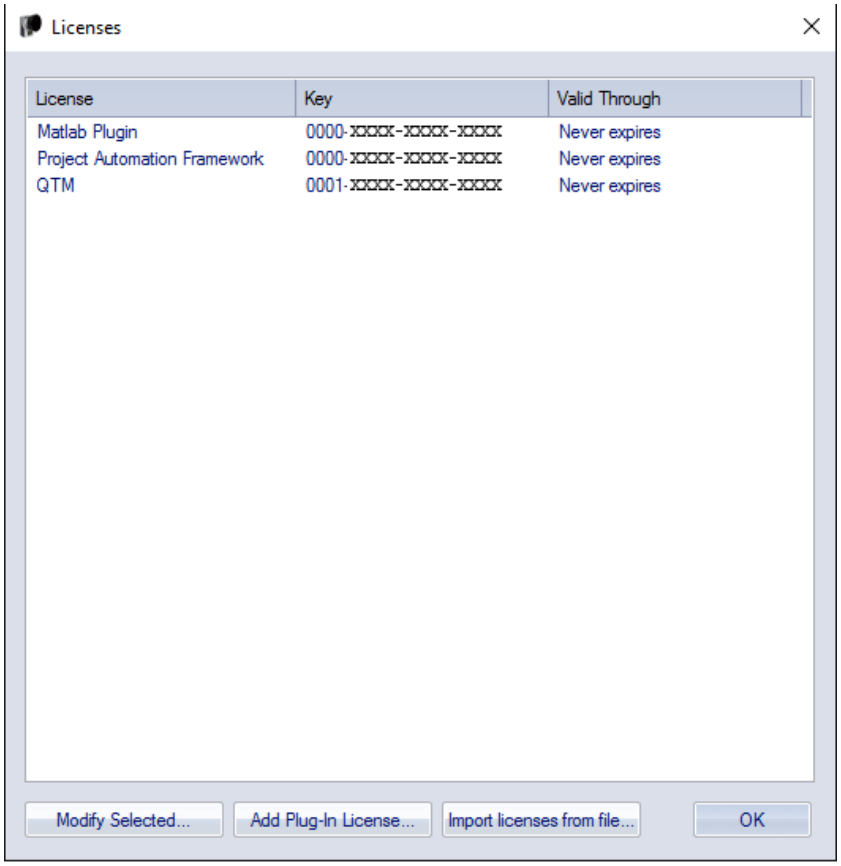


Note: If the license is time limited you must check the **Time limited** checkbox and enter the correct expiration date.

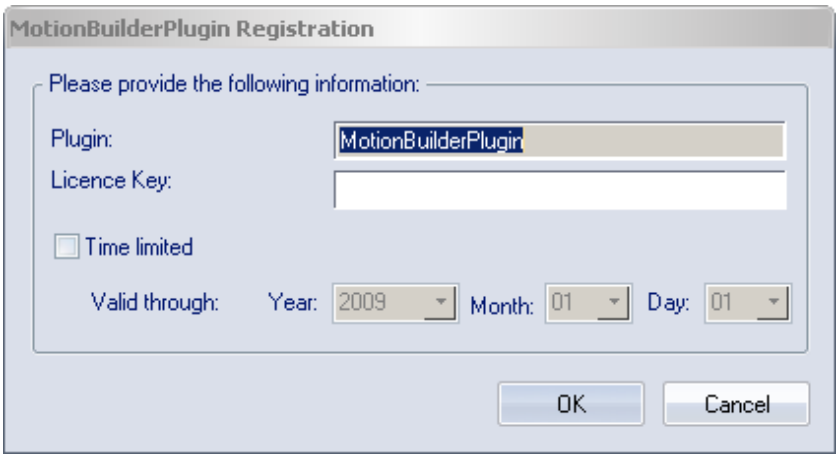
Enter plug-in license

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.



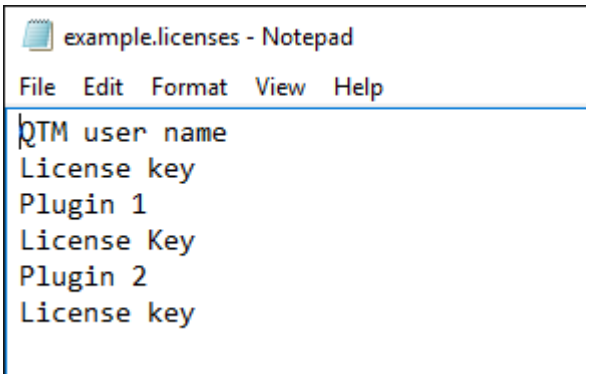
Click on **Add plug-in license** to install a new license.



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.

Alternatively, the licenses can be imported from a text file (*.licenses). The information in the file is organized per row in the following format (replace text with registration data, using exact names for the QTM user name and the plugins, which you can find at http://www.qualisys.com/my/):



Note: File import is not possible for time limited licenses.

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. loading projects. QTM also works if you are just a regular user, but there are some changes compared with being administrator.

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 projects. 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 253.
- Linearization of the cameras, see chapter “Linearization of the cameras” on page 263.

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

- Analog board, see chapter “How to use analog boards” on page 450.
- Force plate, see chapter “How to use force plates” on page 454.
- External trigger, see chapter “How to use external trigger” on page 475.
- Web camera, see chapter “Video capture” on page 464.

Training resources

Besides the QTM manual the following training resources are available.

Getting Started with your Qualisys System
The Getting Started with your Qualisys System guide (PDF) is included in the welcome information for new customers. The Getting Started guide is based on QAcademy tutorials, which can be referred to for more detailed information. An online version of the Getting Started guide is available here.

QAcademy
QAcademy is the official Qualisys online training library, including video tutorials, courses and guides covering a wide range of topics from basic camera setup to advanced data processing. Most video tutorials also include a written manual. A selection of basic tutorials is publicly available. For access to all QAcademy resources, an active support contract is required. QAcademy can be accessed online here.

Documentation
The installation of QTM contains a *Documentation* folder, including several PDF documents, for example, the latest Getting Started guide, a keyboard shortcut reference, and marker set guides for sports and animation. Documentation about the real time protocol of QTM is available in the *RT Protocol* folder.

Store real-time data

The processed real-time data is saved directly to the QTM file when the option **Store real-time data** is enabled on the **Processing** page. This option can for example be used to speed up the capture process and get information if the captured data is good. In this mode only the export processing steps are active and all processing steps that changes data are disabled.

All of the raw data is stored in the file, so that the data can be reprocessed. The data may improve when reprocessing the file, this is especially true for AIM reprocessing which can use all of the data in the file to identify trajectories. The following applies to the different data types:

Data type	Process every frame option enabled	Process every frame option disabled
2D (raw)	All raw 2D data is included in the file	All raw 2D data is included in the file
2D (processed)	All of the 2D data is processed and used for 3D tracking	Some 2D data may not be processed or used for 3D tracking
3D (unidentified)	All of the 3D data is tracked using all 2D data	Some 3D data may not use all of the 2D data available in the file. The result can be 3D data with reduced quality.
3D (identified)	AIM can only use old 3D data to identify the trajectory. When reprocessing the data AIM can use all of the data in the file for identification.	Same as for when the option is enabled and the reduced 3D data quality may also affect AIM .
6DOF	Same as for 3D (identified)	Same as for 3d (identified)
Skeleton	Same as for 3D (identified)	Same as for 3d (identified)
Analog	All of the analog data is included and offsets are applied to all of the analog samples	All of the analog data is included and offsets are applied to all of the analog samples
Force	Forces are calculated for all of the samples	Forces are calculated for all of the samples
EMG	All of the EMG samples are included in the file	All of the EMG samples are included in the file
Eyetracker	Reprocessing is recommended, because the delay between eyetracker and 6DOF data is not compensated for in real-time. The Gaze data is calculated for all of the samples.	Reprocessing is recommended, because the delay between eyetracker and 6DOF data is not compensated for in real-time. Gaze data is calculated for all of the frames that include 6DOF data.
External video	All video frames are included and the time offset is applied to the video.	All video frames are included and the time offset is applied to the video.

Note: When the capture rate is too fast. then the real time frequency is reduced to half of the capture rate or even further reduced. When storing the real-time data it means that 3D data is only processed for all half of the frames, or even less than that for a frequency higher than 400 Hz.

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 or use the keyboard shortcut **Ctrl+N**. 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.

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 following sections describe some typical use scenarios and recommendations on how to use projects. However, projects can be used in many different ways, depending on how you want to organize and share your settings.

Think about the following when deciding how to use the projects.

1. When to create a new project.
    a. It is usually recommended to use one project per marker setup. Then you can save different subject in sub-folders of the project data folder.
    b. You may of course create any amount of projects based on for example the people using the lab.
2. Settings to use when creating a new project, see chapter “Creating a new project” on page 17 and “Project presets” on page 22.
3. It is recommended to save all of the measurement in the project data folder, because then you can browse them in the Project view window, see chapter “Project view” on page 19.
4. Do you want to backup settings, see chapter “Backup of project settings” on page 22.
5. Do you want to view the list of recent projects at startup or always open a certain project, see chapter “QTM startup procedure” on the facing page.

Read the following sections for advice on specific setups: “Projects with qualified users” below, “Projects with students” on the facing page and “Using projects on multiple computers” on the facing page.

Projects with qualified users

For qualified users, projects are helpful to manage the settings of multiple projects or studies. Below follow some suggestions.

1. Create a new project when there is a new study, with a new marker setup.
2. Save all of the QTM files in the project data folder as you can then browse the data in the Project view in QTM, see chapter “Project view” on page 19.
3. Make a backup of the project settings if you like to be sure that you can always go back to the settings you know are correct, see chapter “Backup of project settings” on page 22. If someone opens the project any changes they do to the settings are saved automatically.
    a. If you want to completely protect your projects from other users it must be done via Windows, for example by having multiple logins to the computer.

4. Pin projects that are often used in the **Open project** dialog, see chapter “Manage projects” on page 18.
5. Create a **Project preset** if you create a lot of new projects and want to use the same settings when it is created, see chapter “Project presets” on page 22.

Projects with students

When working with students using QTM, projects can be used as follows.

1. Create a Project preset so that the students can start with the same settings, see chapter “Project presets” on page 22.
2. Tell the students to create a new project with the Project preset and save them at a specific place. The default folder is Documents in Windows, but it can be changed, see chapter “Folder options” on page 250 .
   a. It is a good idea to use a different Windows login for the students so that they cannot access other projects.
   b. If the students can access the other projects make sure that you backup the settings of the projects, see chapter “Backup of project settings” on page 22.
3. Make sure that the QTM files are saved in the project data folder. It is the default path if nothing has been changed in the project settings.

Using projects on multiple computers

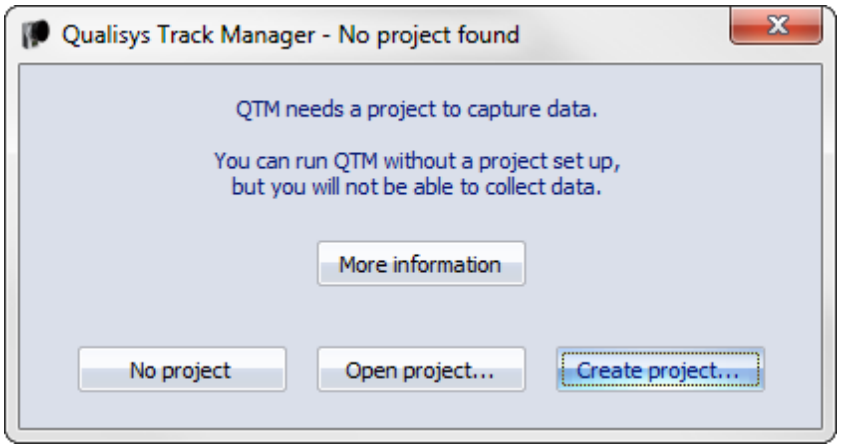
If you use a project with different camera systems you can either move the whole project (or computer) or just move the Data files and AIM models.

- It is recommended to setup the project on both computers and just move the Data and AIM files. Because then there is no risk for losing settings, on the other hand you have to remember to change the settings on both places if you need to.
  - If you move the whole project, then you need to decide on how to handle the camera settings. QTM will automatically detect on **New** if the cameras have changed and you will get the option to import all the settings from another project. It is important to notice that it will overwrite all the settings in the current project, so you will need to add any settings that differs again, for example the AIM model.

    Note: If you locate the system first, QTM will not give a warning for a new camera system, because it is seen as you then wanted to check for a new system.
  - If you just move the Data files there is no problem in opening them on the other computer, because all of the information needed to reprocess the file is included in the file.

QTM startup procedure

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



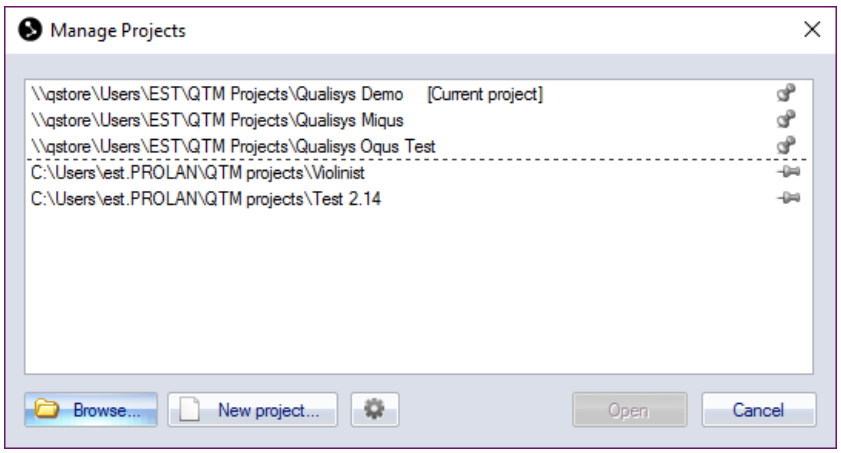
There are three ways to start QTM when you have not used projects before.

Create project
This is the default option because you must have a project to capture data in QTM, see chapter “Creating a new project” on the facing page.

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 or change any project options.

After using projects the first time the default startup procedure for QTM is to display the **Manage projects** dialog. Each time you start QTM you have to select the project to load, to make it less likely that someone make changes in a project by mistake. For more information about the dialog see chapter “Manage projects” on page 18.

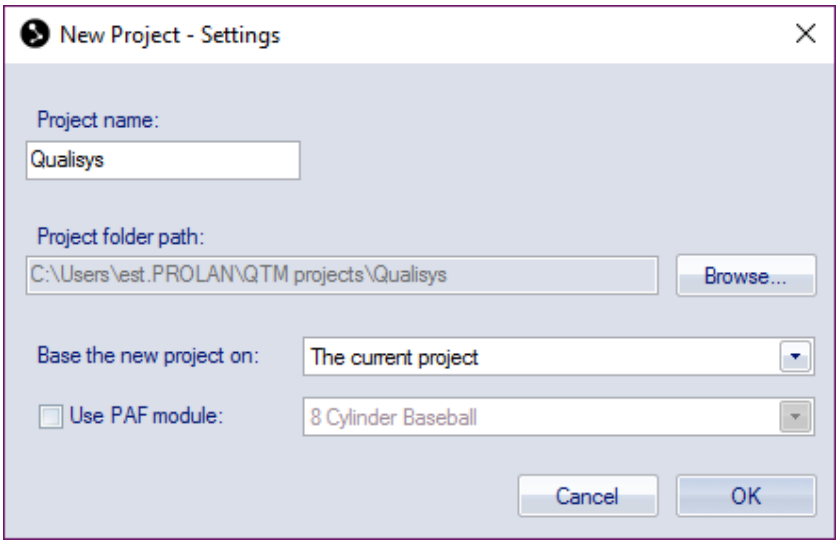


It is also possible to choose to open the most recent project or a selected project when starting QTM. To set these click the button, see chapter “Startup” on page 251 .

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**.

Note: To change the project name after it has been created use the **Rename Project** option on the **File** menu.

Project folder path

This is the path of the project folder. The default is the Documents folder in Windows. Click **Browse** to change the path.

Note: The **Default path for new projects** can be set on the **Folder options** page in the **Project options** dialog.

Note: 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.

Base the new project on:

Select the source of the settings that are used when creating the project with the drop-down box.

Custom preset: ...

Any project presets available on the computer are displayed at the top of the list, for more information about the presets see chapter “Project presets” on page 22.

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 current settings

The settings are copied from current settings in **Project options**. This is the default option when creating a project with the **New project** option on the **File** menu.

Note: There is no current project when you start QTM therefore the default option is then **Settings imported from another project**.

Settings imported from another project
Browse for the project that you want to base the new project on and open the project file.

Default settings
All settings are set to the default values and all of the current settings are deleted.

Use PAF module
Check if you want to create a project with project automation. QTM includes example projects for Excel, Matlab and Python. If you have PAF analysis modules installed on your computer, they will show up in the list. For more information about PAF, see “Appendix G: General PAF information”

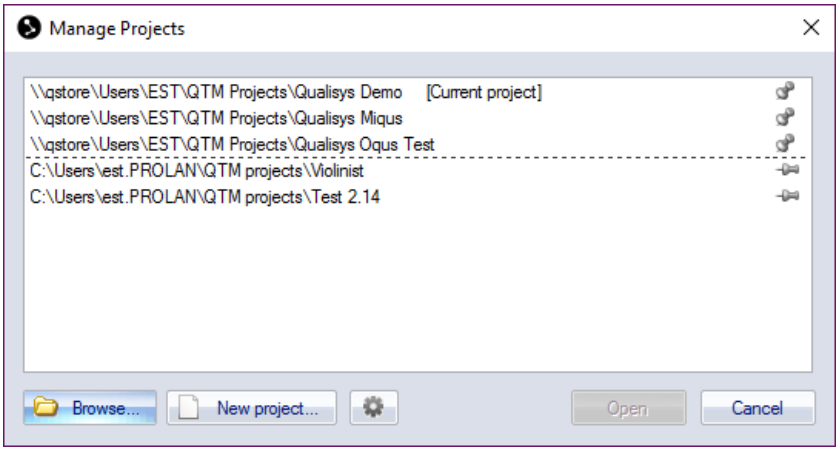
Note: If you select any of the options other than the **Current settings**, you will lose the current settings in **Project options**. If you haven't got any projects in the recent projects list, then the settings are saved in a backup in C:\ProgramData\Qualisys\Workspace backups.

Manage projects

Use **Manage projects** on the **File** menu to manage the projects, for example to switch to a recently used project or open another project. When switching project all of the currently open files will be closed and you have to save any unsaved files. Then the settings in **Project options** and other settings such as those in the **Start capture** dialog are loaded in QTM from the new project.

Note: 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 **Manage projects** dialog displays the 100 most recently used projects. Double-click on a project to open it. You can pin projects so that they are always on the top of the list, you have to click **Open** for the pinning to change.
The current project is shown when with **[Current project]** next to the name. The current

project is also displayed in the title bar of QTM and **Project options**.

Note: When QTM starts up there is no current project, since it has not been loaded yet.

The following options are available in the dialog:

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

New project
Create a new project, see chapter “Creating a new project” on page 17.

Open settings for the startup of QTM, see chapter “Startup” on page 251 .

Open
Open the project selected in the list above.

Project view

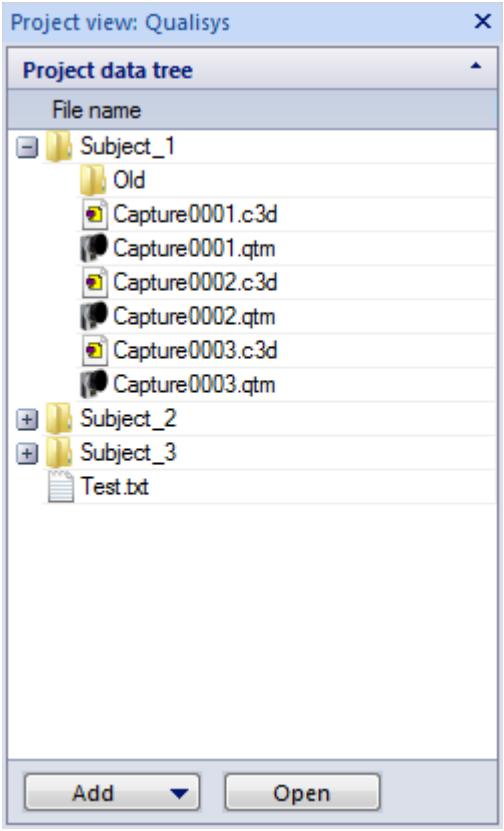
The **Project view** window displays the data files included in the current project. The view consists of two parts: **Project data tree** and **Details** . The name of the current project is displayed at the top of the Project view.

To open the **Project view** window go to **View** menu and select **Project view**, the shortcut to toggle the window is ctrl+r.

Project data tree

The **Project data tree** window is used to display and open the files in the Data folder of the current project. This includes folders and QTM files, but also all other files. You can drag and drop files to the Project data tree from for example Windows explorer. If the file is dragged from another folder on the same hard drive it is moved to Data folder. You can toggle whether to copy or move the file with the CTRL key. If the file is on another hard drive or the network then a new copy is made in the Data folder.

The data tree can be searched with **Find** (Ctrl+F) on the **Edit** menu. When searching the first occurrence of the search term will be highlighted in the data tree. Use **Find Next** (F3) to search for the next occurrence of the current search term.

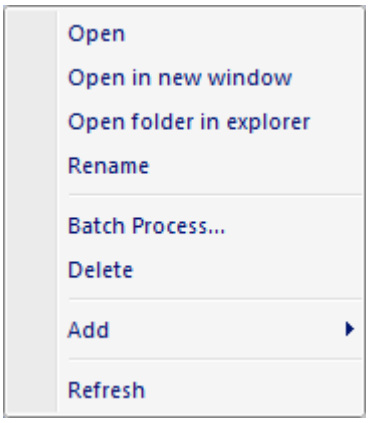


There are two commands at the bottom of the data tree: **Add** (**New folder** or **New measurement**) and **Open**.

When adding a folder it is placed in the currently selected folder, if none is selected it is placed in the root of the Data folder. Adding a new measurement is the same as starting a new capture with the **Capture** command on the **Capture** menu and the folder to save the file in is set to the currently selected folder in the project view.

The **Open** command is only available when a file is selected and it will then open that file. If it is a QTM file it is opened in the current QTM window or a new window, depending on the current settings on the **GUI** page in **Project options**. Other files are opened with its corresponding Windows program.

The QTM file that is open in the current QTM window is displayed in bold. If you open a file that is already open in another QTM window you will switch to that window.



Right-click on a file or a folder to get the following options.

Open (only available for files)
Opens the selected file in QTM if it is a QTM file. Otherwise it is opened by its corresponding Windows program.

Open in New window (only available for QTM files)
Opens the QTM file in a new QTM window. It overrides the **Close current file** option on the **GUI** page.

Open folder in explorer
Open the currently selected folder in Windows explorer so that you can for example move the files.

Rename
Rename the selected file or folder.

Batch Process...
Batch process the currently selected files. It will open the **Batch processing** dialog so that you can select which processing steps that you want to apply, for more information about batch processing see chapter “Batch processing” on page 329.

Delete
Delete the selected file or folder.

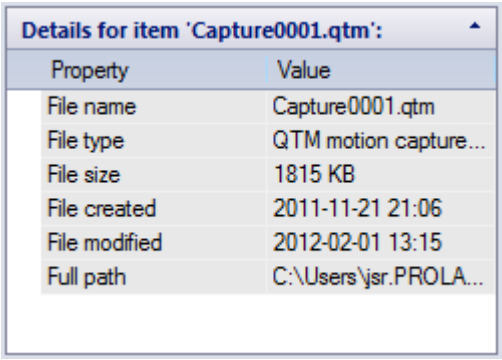
Add

New folder
Add a new folder in currently selected folder. If no folder is selected it is placed in the root of the Data folder.

New measurement
Add a new measurement, which is the same as **Capture** on the **Capture** menu and the folder to save the file in is set to the currently selected folder in the project view.

Refresh
Refresh the content in the **Project data tree** so that it matches the content of the project data folder. This is usually done automatically by QTM.

Details



The **Details** window displays information about the currently selected file. The displayed data is:

File name

File type

File size

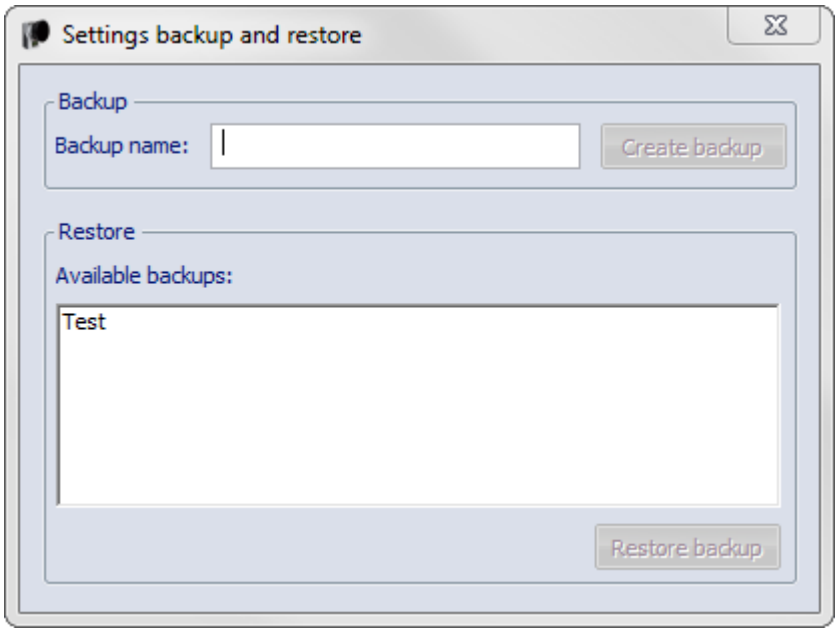
File created
This is not the time when the data was captured, but when the file was created on the computer which is not the same if the file has been copied.

File modified

Full path

Backup of project settings

The project settings can be saved in a backup file. The backup can then be used if you want to use two different sets of settings in the same project or if you want to be sure that the settings are not changed by someone else, because the project settings are saved automatically when they are changed in QTM. Click on **Settings backup and restore** in the **File** menu to create and 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.

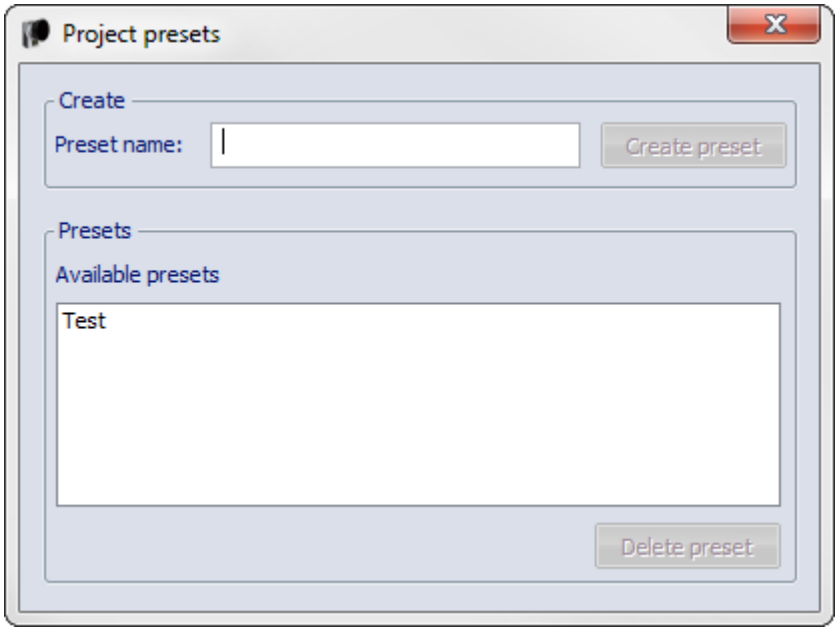
Note: QTM will use the latest calibration made on the computer, that was made with the same cameras (placed in the same order). It means that the calibration may be loaded from another project when switching projects, it is then copied to the Calibrations folder in the project folder.

Project presets

The project presets can be used when creating projects to make sure that you start with the same settings. The project preset contains all of the settings QTM (Project options and other settings such as those in the Start capture dialog). The preset also contains any AIM model that was in the project AIM folder when creating the preset.

Note: The presets are saved globally, in the folder

C:\Users\????\AppData\Roaming\Qualisys\Project presets\, so that it can always be accessed when creating projects in QTM.



To create a preset follow these steps.

1. Open a project with the settings and AIM models that you want to use.
2. Go to the **File** menu and select **Settings management**/**Project presets** to open the **Project presets** dialog.
3. Enter the name of the preset and click on **Create preset**.

To use a preset follow these steps.

1. Go to the **File** menu and select **New project**.
2. Enter a **Project name** and select the settings to **Base the new project on** from the drop-down list. The presets are listed as **Custom preset:** followed by the name.

If you want to change any settings in a preset you need to create a project with the preset and then change the settings. Then create the preset again with the same name.

A preset can be deleted from the **Project presets** dialog. It is important to notice that the presets are deleted from the computer so that it can no longer be accessed by anyone.

Project folder

The project folder is used for organizing a QTM project. The folder is displayed in Windows explorer with a special icon to indicate that it is a QTM project folder. It contains the following files and folders:

Data
This is the default 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.

Meshes
This folder contains all of the meshes associated with the current project.

Messages
This is the folder for the messages log files.

Settings
This folder contains the backups of project settings.

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

Note: The project file may change format with a new version of QTM. A backup is saved of the previous version of the file, it can be named for example Settings.qtmproj.ver_100-101.

Note: The project file in before QTM 2.9 was called settings.qps. A backup (.bak) is saved of this file when it is converted to the qtmproj format.

QTM creates a default project folder called Qualisys in My Documents 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.

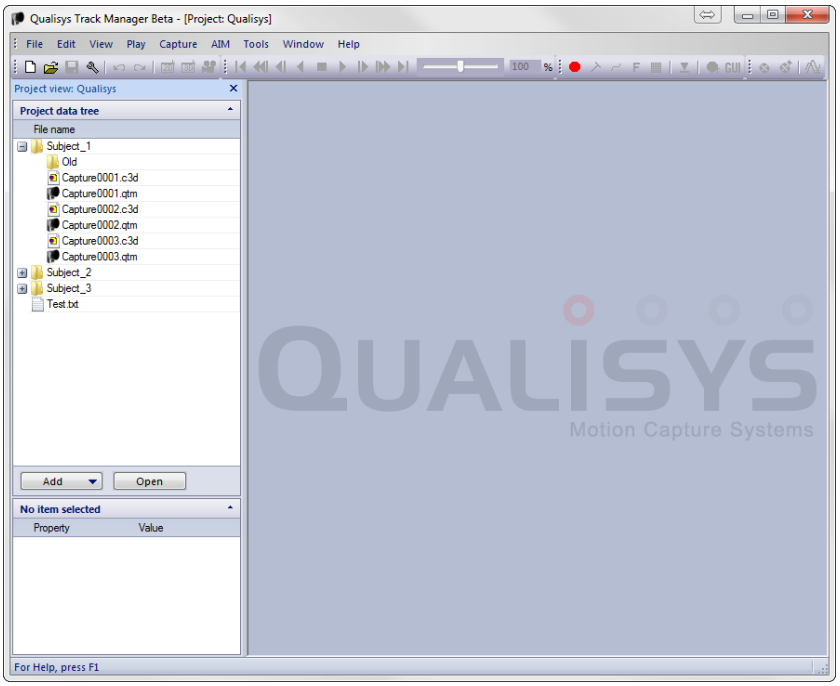
There are other settings and files which are needed for all users that use QTM on a computer. To see where these are saved check the **Folder options** page in the **Project options**, see chapter “Folder options” on page 250.

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 64-bit operating systems).

QTM windows

Main window

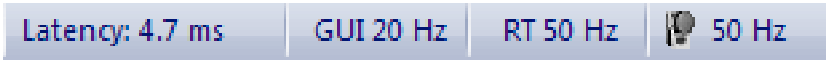
The main window is opened when QTM is started. It contains the different menus and toolbars, see chapters “Menus” on page 101 and “Toolbars” on page 109. At the bottom of the window there is a **Status** bar, which shows important messages and progress information. On the left side there is a Project view which displays the files included in the current project, see chapter “Project view” on page 19.



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 269.



Time code
Displays current value of incoming time code (SMPTE, IRIG, Camera time).

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 315.

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.

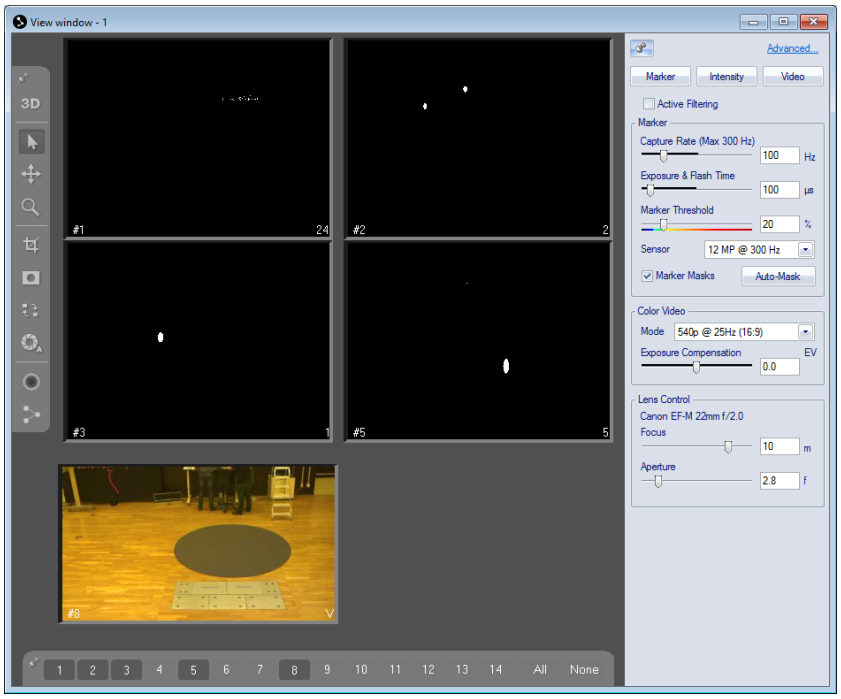
View windows

In a View window 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 59.

2D view window



The 2D view window contains 2D motion capture data or images of the cameras connected to QTM. The different types of data that can be displayed are:

- 2D motion capture data of Qualisys cameras in **Marker mode**,
- Colored intensity maps of Qualisys cameras in **Intensity mode**, or
- Video images of Qualisys cameras in **Video mode** or external video devices.

The 2D view window includes the following controls:

2D view toolbar
Contains tools for the 2D view, see chapter “2D view toolbar” on page 29.

Camera selection bar
Select the cameras that you want to view in the 2D view window, see chapter “Modifying the 2D view” on page 30.

Camera settings sidebar
Control the basic settings of the cameras, see chapter “Camera settings sidebar” on page 31.

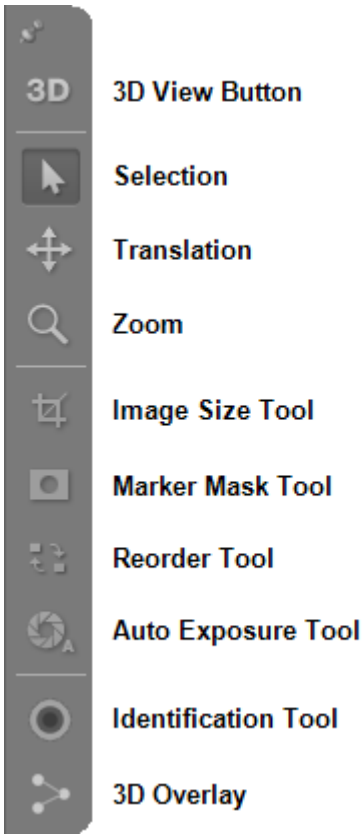
2D view window menu
Right-click to open options for a single camera, see chapter “2D view window menu” on page 39.

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.
- After the camera number, the type of camera is indicated for Qualisys cameras.
- When delayed exposure is enabled, the exposure group number is displayed after the camera type, e.g. **(exp. group: 1)** for cameras in exposure group 1.
- 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 most types of Qualisys cameras the **Image size** can be changed with the **Image size** tool on the **2D view** toolbar, see chapter “2D view toolbar” on the facing page.
  Note: In a file, only the active part of the sensor is displayed. I.e. if the **Image size** has been reduced, then the aspect ratio of that 2D view will match the reduced image size.
- Camera marker masks are shown as green squares, see chapter “How to use marker masking” on page 297.
- Software marker masks are shown as blue squares, see chapter “How to use software marker masks” on page 331.
- Position and size of detected markers for Qualisys cameras in marker mode. The default color of the detected markers is white. In case rays are displayed the colors of the markers correspond to the associated 3D trajectories.
- The center of the 2D markers is indicated by a crosshair.
- The image from Qualisys cameras in video mode 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 using Qualisys cameras is described in the chapter “Qualisys video capture” on page 286.
  Note: In a file, only the active part of the video image is displayed. If the **Image size** has been reduced, then the aspect ratio of that video view will match the reduced image size.
- The external video devices 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 video devices see chapter “External video devices in 2D view” on page 36.
- The 2D markers are color coded when marker filtering is activated (only available for Oqus cameras), see chapter “How to use non-circular marker settings” on page 299.

- 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**.
- 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.

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.

3D View Button
Switch to 3D view.

Selection
Use the normal mouse behavior.

Translation
Use the left mouse button to translate the 2D view.

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

Note: You can also use the mouse wheel for zooming.

Image Size Tool
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 130.

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.

Note: When showing a marker view or video image in a file, only the active part of the marker view or video image is displayed. I.e. if the **Image size** of the camera has been reduced, then the aspect ratio of that view will match the reduced image size.

Marker Mask Tool
Use the left mouse button to draw a new marker mask for a camera. Marker masks can either be added while in preview mode (**Marker** or **Marker intensity**) and is then added to the list of camera masks, or when added to a camera in a file then the masks are only used in reprocessing of that file.

A new camera marker mask will be added as a green area and the markers inside the mask will be removed immediately, see chapter “Marker masking” on page 297.

A new software marker mask is added as a blue area and the 2D data is not affected until the file is reprocessed, see chapter “How to use software marker masks” on page 331.

Reorder Tool
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.

Auto Exposure Tool (only available for Miqus Video and Oqus 2c cameras)
Draw the Auto exposure area with this tool. The area is displayed as a gray rectangle when the tool is active. By default the area is maximized to the current image size, but if for example there is a very bright part of the image the auto exposure will work better if the area is reduced. For Miqus Video Color the auto exposure area is also used to set the white balance.

Identification Tool
Click to switch on the green LED ring on the selected Arqus or Miqus cameras.

3D Overlay
Click to turn on/off 3D overlay for the selected cameras.

Modifying the 2D view

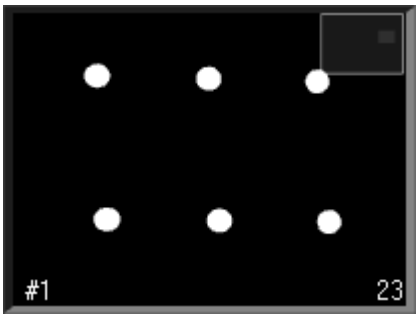
The appearance of the 2D view can be modified with the following options.

- 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 camera settings for the Qualisys cameras. It is displayed at the right side of the 2D view window when the camera system is live. Use these settings when setting up the camera system to get the best data. The settings are also available on the **Cameras** page in the **Project options** dialog, refer to chapter “Cameras” on page 127 for more details about the settings.

The sidebar is pinned by default, so that it is always visible. When unpinned, it slides out when you move the mouse on the edge to the right in the 2D view window.

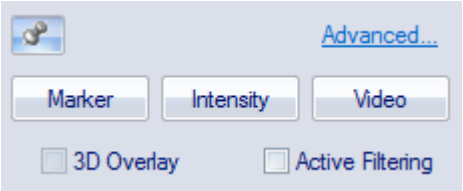
The sidebar will only display settings that are available for the currently visible (selected) cameras. For example, when all cameras are in marker mode the **Video** settings are hidden. All of the settings, except the **Marker Capture Rate**, apply only to the currently visible cameras in the 2D view 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 **Marker** and **Video** settings apply to all marker cameras models, as well as Oqus high-speed video cameras. The settings of the streaming video cameras (Miqus Video and Oqus 2c) are available under **Streaming Video**.

The following settings are available on the sidebar.

Camera Mode



These settings change the mode of the camera and you can also activate some other options.

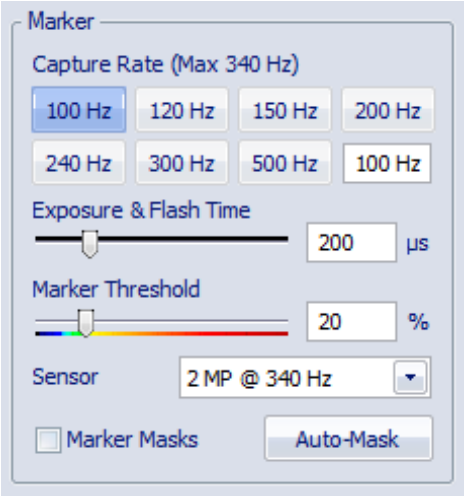
Marker, **Intensity** and **Video**
Switch between the different camera modes. For a description of the different camera modes see chapter “Video preview in QTM” on page 292.
The mode can also be changed individually on a camera from the **2D view window** menu.

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 (not available for Oqus 3 and 5 series and Miqus)
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 296.
Note: You can still turn on active filtering for the supported models, even if there are other camera models in the system.

Advanced...
Clicking the **Advanced…** link will open the **Cameras** page in the **Project Options**. The currently selected cameras will be selected in the camera list.

Marker



These settings change marker settings for the visible cameras.
Note: The **Capture Rate** applies to all cameras.

Capture Rate
The capture rate that is used by cameras measuring markers. The maximum capture rate shown above the capture rate buttons applies to the system at full FOV. Buttons with a value exceeding the maximum capture rate are indicated in red.

Note: If the camera system includes different camera types then the dark blue bar indicates the capture rate for the camera type with the lowest limit for the maximum image size.

Note: The image size of 2c-, 5+- and 7+-series cameras is only limited in the y-direction when increasing the capture rate. However, the image ratio is kept constant when increasing the capture rate. You can increase the width of the image manually by dragging the sides of the red rectangle.

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 258. The current maximum **Exposure and Flash Time** is displayed with a dark blue bar.

Note: If the camera system includes different camera 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 17. 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 258.
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.

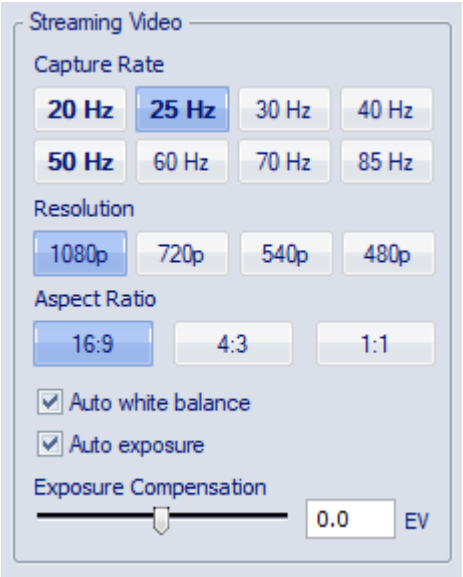
Marker Masks
Toggle the marker masks. If deselected the marker masks in the current cameras will not be used by the camera. The masks will be grey and the markers below them will appear.

Auto-Mask
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 298.

Note: The number of masks per camera is limited to 20 for Arqus and Miqus cameras and 5 for Oqus cameras. If there are more unwanted reflections you will need to manually edit the masks.

Sensor Mode
Switch Sensor mode for the current cameras in marker mode. You can select between a full size mode and high speed sensor modes. Use the high speed modes for example if you need to capture at higher frequencies and still want the full FOV, but you do not need the full resolution. For an overview of available sensor modes per camera type, refer to the table in "Qualisys camera sensor specifications (marker mode)" on page A - 1.

Streaming Video



These settings change streaming video settings for the visible streaming video cameras (Miqus Video or Oqus 2c).

Capture Rate
The capture rate that is used by cameras capturing video. The video capture rate can be set by pressing one of the available buttons. The buttons show commonly used values for video capture rate. Integer divisions or multiples of the current marker capture rate are indicated bold. The number of buttons and their exact values depend on the current maximum video capture rate. The maximum capture rate for the current settings (resolution and aspect ratio) is displayed above the buttons.

Resolution
Set the resolution of the video image by pressing one of the four buttons. The available values are 1080p, 720p, 540p and 480p (the values indicate the vertical dimension of the image in pixels).

Aspect ratio
Set the aspect ratio of the video image by pressing one of the three buttons. The available aspect ratios are 16:9, 4:3 and 1:1.

Auto Exposure
Check to use automatic exposure for streaming video cameras. When activated the **Exposure Time** and **Gain** options are hidden and controlled by the auto exposure. Optionally, use the **Auto Exposure Tool** to limit the image area used to set the exposure (see chapter “2D view toolbar” on page 29).

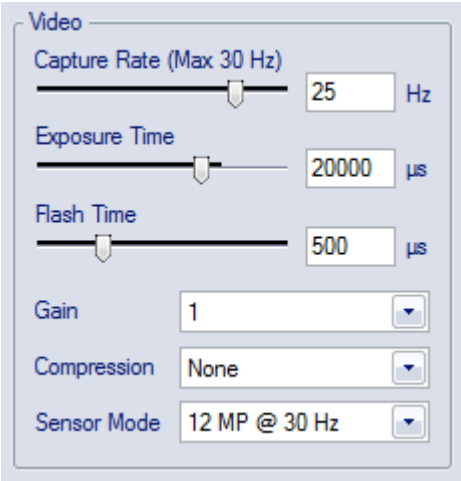
Auto white balance
Check to use automatic white balance. This option is only available for the Miqus Video Color camera. Optionally, use the **Auto Exposure Tool** to limit the image area used to set the white balance (see chapter “2D view toolbar” on page 29).

Exposure Compensation (auto exposure only)
The **Exposure Compensation** option can be used to adjust the exposure of the image when using **Auto Exposure**. Increase the value (positive EV) for a brighter image or decrease the value (negative EV) for a darker image.

Exposure Time (manual exposure only)
The exposure time used by the selected streaming video cameras. The current maximum **Exposure time** is displayed with a dark blue bar.

Gain (manual exposure only)
The **Gain** option can be used to change the sensitivity of the sensor. Increase the gain for a brighter image. The available values are 1, 2, 4 (default), 8 and 16. Note that the image quality may decrease at high gain values.

Video



These settings change video settings for the visible cameras.

Capture Rate
The capture rate that is used by cameras capturing video. The maximum capture rate for full FOV is displayed over the slider.

Note: If the camera system includes different camera types then the dark blue bar indicates the capture rate for the camera 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. If the image size has been set manually for a camera the reduced image size will have the same relations for x and y.

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 290. The current maximum **Exposure time** is displayed with a dark blue bar.

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 Off (0) 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.

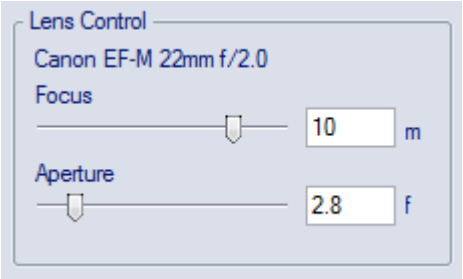
Gain
Set the gain for the current cameras in video mode to get a brighter video

image. Depending on the camera type you can use gain values of 1, 2 or 4 and for some cameras also 8 and 16.

Compression
The Compression setting can be used to switch between **None**, **In-camera MJPEG** and **Software** compression. The default for Oqus 2c, 5+, and 7+ is **In-camera MJPEG**, which is the recommended setting for those cameras. For the other cameras the default is **None**, however, most of the time it is recommended to select **Software** and a Codec to reduce the video file size.

Sensor Mode
Switch sensor mode for the current cameras in video mode. You can select between a full size mode and high speed sensor modes. Use the modes for example if you need to capture at higher frequencies and still want the full FOV, but you do not need the full resolution. For an overview of available sensor modes per camera type, refer to the tables in “Oqus high-speed video sensor and memory specifications” on page A - 2 and “Qualisys video sensor specifications (in-camera MJPEG)” on page A - 3.

Lens Control



The type of lens on the camera is shown above the settings.

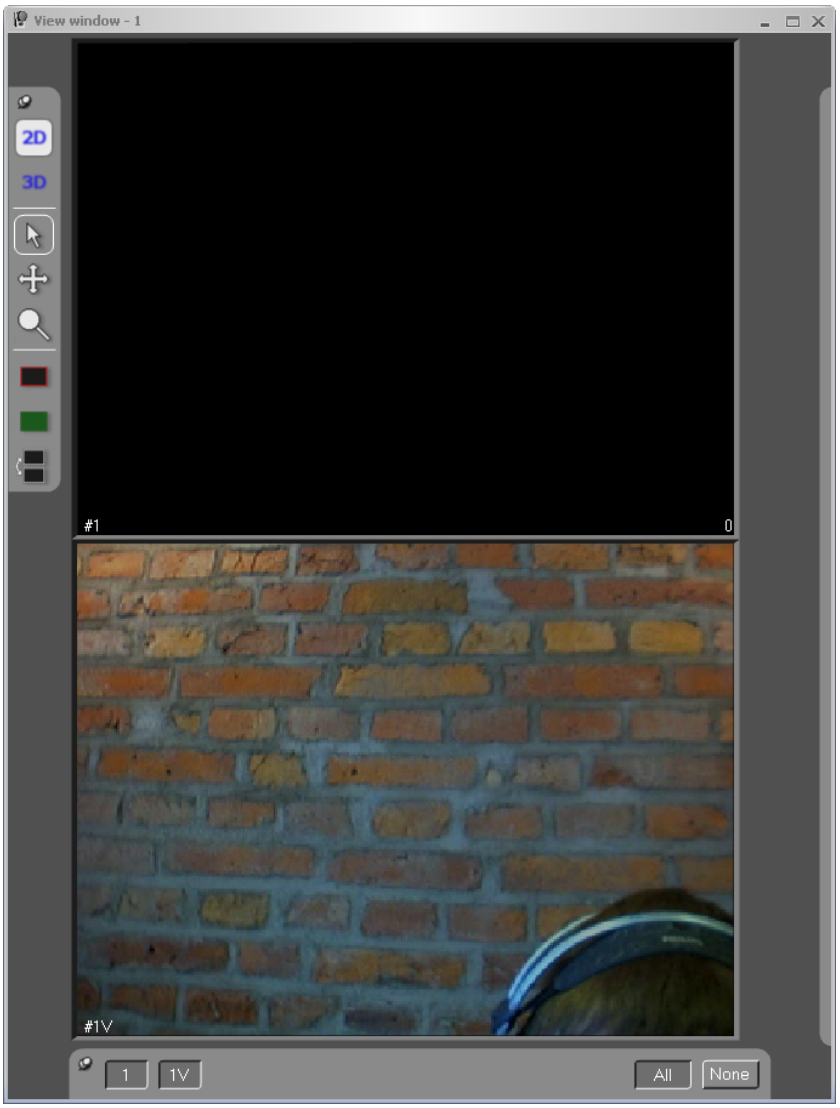
Focus (available for 7+ series with a motorized lens)
Set the focus for the current cameras. The same setting is used for marker and video mode. The minimum distance is 1 m and furthest distance is 20 m.

Aperture (available for 7+ series with a motorized lens)
Set the aperture for the current cameras. The same setting is used for marker and video mode. The available aperture values are dependent on the lens in the camera.

Note: Once focus and aperture have been set for the cameras in a fixed camera system it is possible to disable lens control using the Qualisys Firmware Installer. This way the current lens settings will be fixed. For more information, see “Appendix B: Qualisys Firmware Installer (QFI)”.

External video devices in 2D view

The external video devices, such as video from Blackmagic cards or web cameras, are displayed last in the 2D view window. The video data 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.

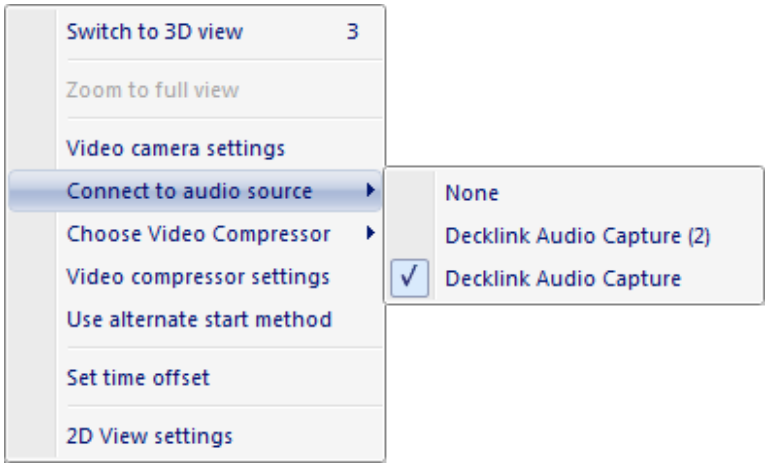
Note: You can have one video camera per 2D view window if you like. Use the camera buttons at the bottom of the 2D view window to select which cameras to view.

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 video devices” on page 464.

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 video camera. For more information,see "How to use video devices" on page 464 or the manual of the camera for camera-specific information.
Note: Only available in preview mode.

Connect to audio source
Choose a DirectShow audio source that is associated with the video camera. For more information see chapter "Selecting audio source" on page 474.
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 473.
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.

Use alternate 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.

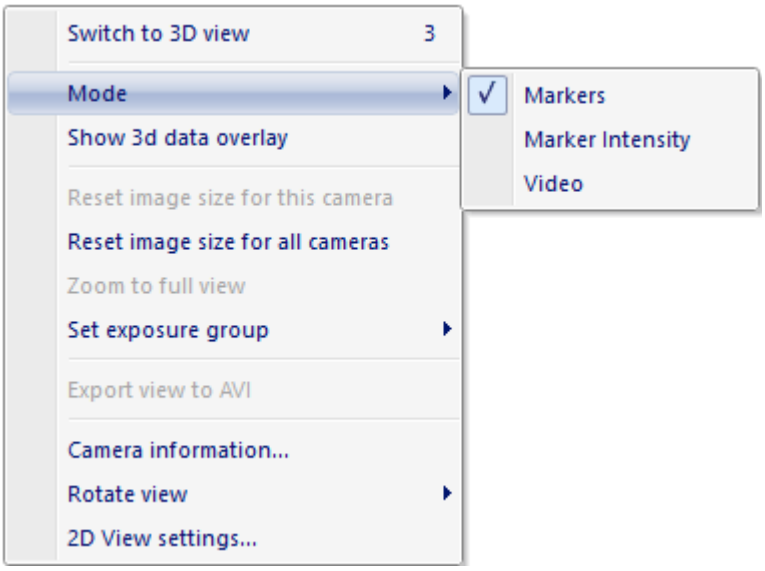
Set time offset ...
Set the time offset between the video file and marker data, see chapter "Video offset" on page 473.

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, see chapter "External video devices in 2D view" on page 36.



The following options are available in the 2D view:

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

Mode

For a description of the different camera modes see chapter "Video preview in QTM" on page 292.

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.

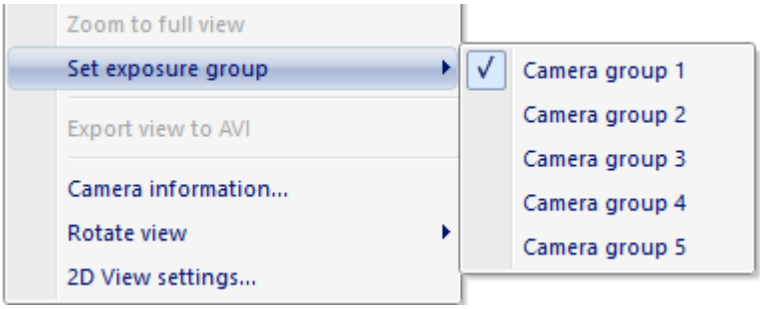
Show 3D data overlay
Overlay the 3D data on the 2D view. This can be done on any camera independent of whether it is **Marker** or **Video** mode. For more information see chapter "3D data overlay" on page 41.

Reset image size for this camera
Reset to full image size for a camera with reduced image size.

Reset image size for all cameras
Reset all cameras to full image size.

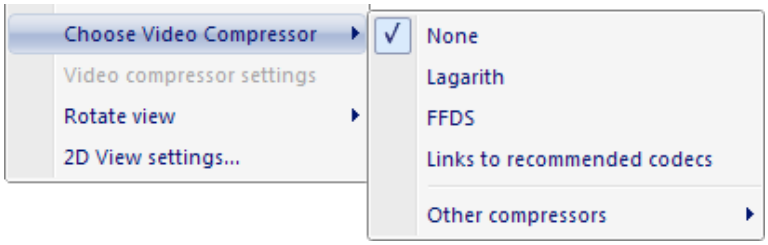
Note: The Reset image size options are only available when QTM is in preview mode before you start a capture.

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.



Set exposure group
Select the exposure group for the camera view that you right-clicked on, see chapter “Delayed exposure to reduce reflections from other cameras” on page 259.
Note: The option is only available when the **Exposure delay mode** called **Camera groups** is selected on the **Cameras** page in **Project options**,

Export view to AVI
Export the camera view that you right-clicked on to an AVI file, see chapter “Export to AVI file” on page 404.

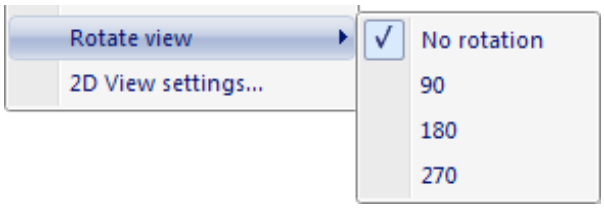


Choose video compressor (Oqus)
Choose a video codec from a list of the installed codecs on the computer. The compressor is selected for all Oqus cameras in Video mode. For more information see chapter “Video compression”.
Note: Only available if the Oqus camera is Video preview mode.

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.

Camera information
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 243.

3D data overlay

The 3D data can be overlayed on the 2D image area 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.

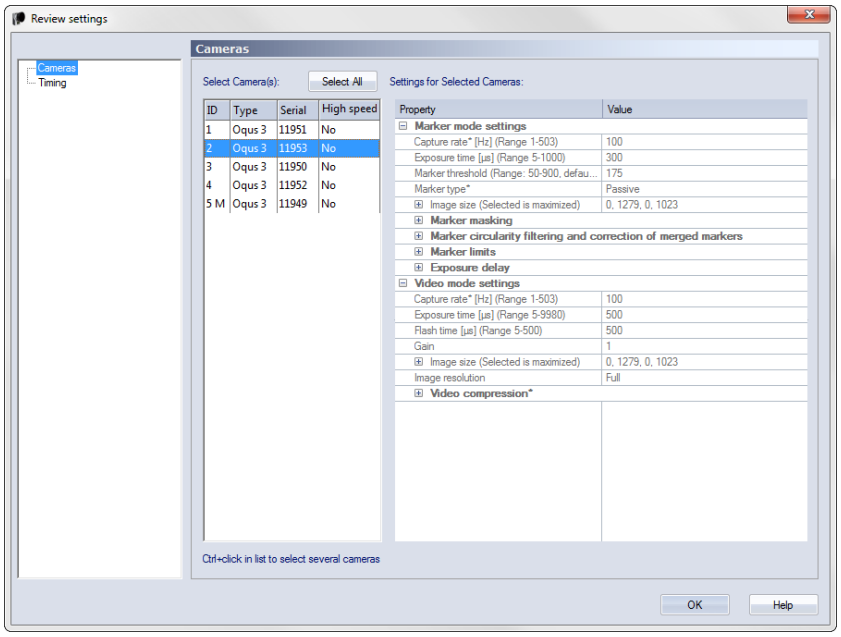
- Calibrate the camera system, including the Qualisys cameras that are used in video mode.
- Open a **2D view** window in RT/preview or a file.
- Right-click on the camera where you want to turn on the overlay and select **Show 3D data overlay**.

  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 243.

  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.

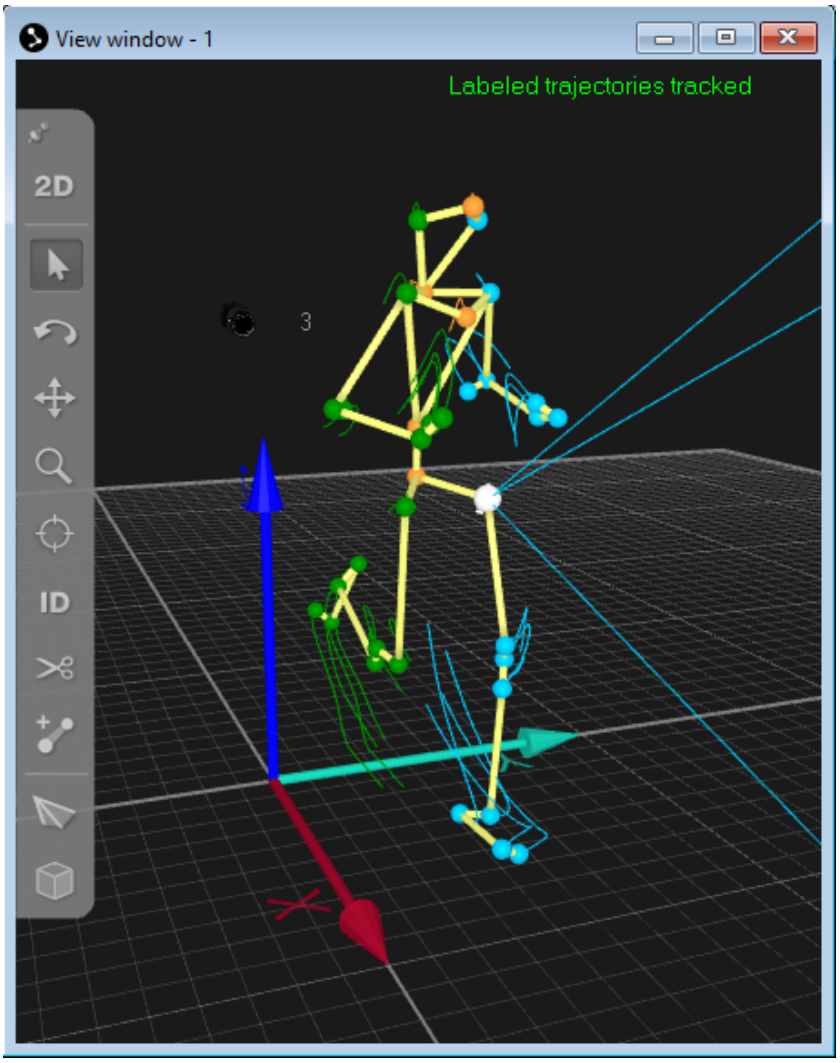
Review camera settings

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 **Cameras** page and **Synchronization** 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 “Cameras” on page 127 and “Synchronization” on page 154.

3D view window



In 3D view window 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 next 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 46.

3D view window menu
Options for the 3D view, opened by right-clicking in the 3D view window, see chapter “3D view window menu” on page 57.

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

- Axes of the coordinate system of the motion capture (X = red, Y = cyan and Z = dark blue).
- A grid showing 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.

- Trajectory count showing the number of selected trajectories and the total number of trajectories in the current frame.
- Markers for the current position of the trajectories, see chapter “Trajectories in 3D views” on page 48.
- Traces of the trajectories.
  Note: Only available in file mode.
- Rays showing which cameras contributed to the selected trajectories, see chapter “Rays in 3D views” on page 52.
- Bones between markers, see chapter “Bones in 3D views” on page 49.
- 6DOF bodies, see chapter “6DOF bodies in 3D views” on page 50.
- Skeletons, see chapter “Skeletons in 3D views” on page 51.
- Rectangular areas showing the location of the force plates.
- Force vectors (red) displaying the current force on the force plate, see chapter “Viewing force data” on page 386.
- 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 53.
- Covered and calibrated volumes, see chapter “Volumes in 3D views” on page 54.
- A white outlined box for the bounding box used by the 3D tracker, see chapter “Bounding box restricting 3D data” on page 186.

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 244.

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 page 46.

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 the following chapters.

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 indicating the center of the 3D view. By double clicking anywhere in the background, the center is automatically moved to the geometrical center of all 3D points at the current frame.

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

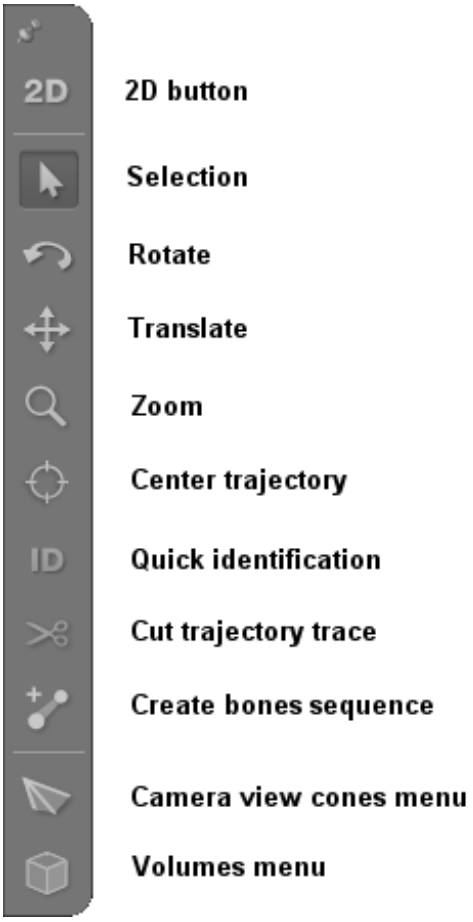
Rotate the 3D view
Hold down the left mouse button in the 3D view window and move the mouse, the view is rotated freely around the red crosshair. Hold down Ctrl + Shift to limit the rotation to the current plane of view.

Translate the 3D view
Hold down the right mouse button in the 3D view window 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 3D view 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.

In addition, it is possible to follow the movement of one or more selected markers by enabling the **Follow Selected Markers** function. This function can be enabled by pressing the **Follow Selected Markers** button in the **Gui Control toolbar** or the keyboard shortcut **Ctrl + Alt + 0**. When this function is enabled, the center of the 3D view will be continuously translated to center of the geometric center of the selected marker(s).

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 362. 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.

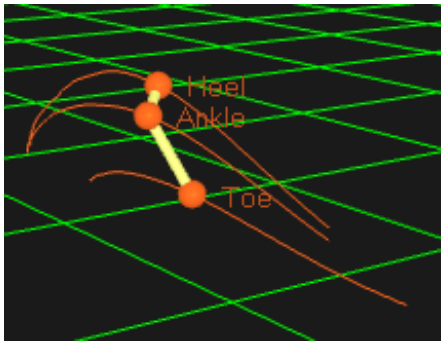
Cut trajectory trace cursor
Click on a trace to cut it in two parts. This is the same as using the Shift + 'x' key with the **Selection** cursor.
Note: The gap between the two parts is there to visualize that there are two parts it is not a missing frame.

Create bones sequence cursor
Click on trajectories in sequence in the 3D view to create bones between them. This is the same as using the Shift + 'b' key with the **Selection** cursor. Just click in the background to restart a new series of trajectories.

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 53.

Volume vizualizations menu
Click on the icon to open the **Volumes** menu, for more information see chapter “Volumes in 3D views” on page 54.

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 63.

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.

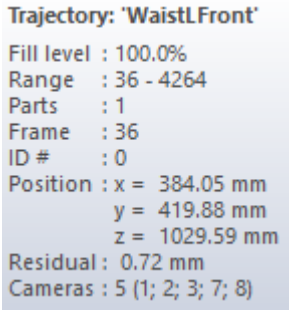
Hold down **Shift** and **drag** the mouse to use area selection.

Drag and drop for identification of trajectories
Trajectories can be drag-and-dropped from the 3D view to labeled trajectories to change their identity. For example, drop an unidentified trajectory from the 3D view onto an empty label in the **Labeled trajectories** window to identify it. In case the labeled trajectory is not empty, this requires that there is no overlap between the trajectories.

A trajectory can also be dropped on the trace of another trajectory in the 3D view to join them. This can be used in combination with scrubbing (hold down **Control** and **drag** the mouse in empty 3D space) and trace range zooming (hold down **Shift** and roll **mouse wheel**).

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 68.

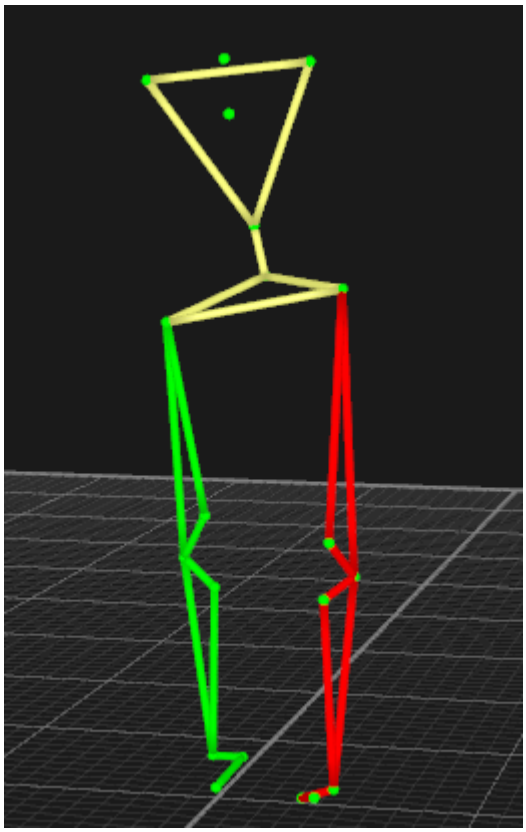
Information
Place the cursor over a marker or a trace to see a tool-tip with information about the trajectory.



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.

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 can have different colors and the bone colors that are used are saved with the AIM model.

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 by using the **Create bones sequence** tool in the 3D view toolbar. The bones will then be created between the bones in the order that you click on

them.

Note: If bones are included in the capture file that is used to generate an **AIM** model, the bones and their colors 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.

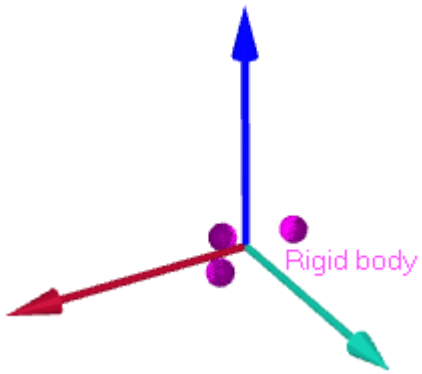
Right-click on the bone and select **Change bone color** to select a color. You can also select multiple bones with Shift + click to change colors on several bones. For other options on the right-click menu see chapter “Bone menu” on page 58.

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

The bones visualization settings are set on the **3D view settings** page in the **Project options** dialog.

Note: You can use the **Bones** button on the **GUI Control** toolbar to toggle the display of the bones.

6DOF bodies in 3D views



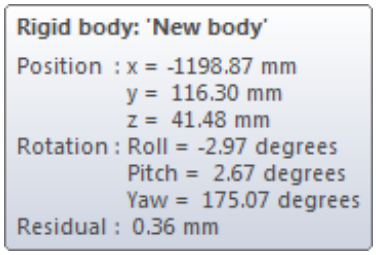
Measured 6DOF bodies are displayed as separate local coordinate systems in a **3D view** window, where the origin is placed at the position of the local origin of the 6DOF body. The axes of the local coordinate system have the same color codes as the coordinate system of the motion capture.

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.

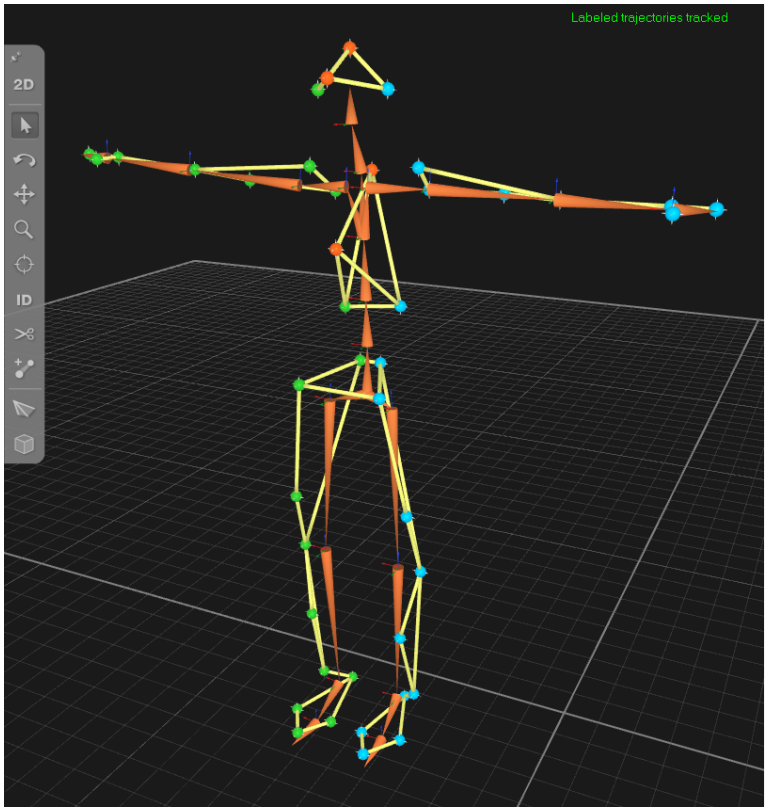
The 6DOF data can be viewed in the **Data info** window or exported to a TSV file and Matlab, see chapter “6DOF data information” on page 87 respectively chapters “Export to TSV format” on page 389 and “Export to MAT format” on page 399.

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 327.

Skeletons in 3D views



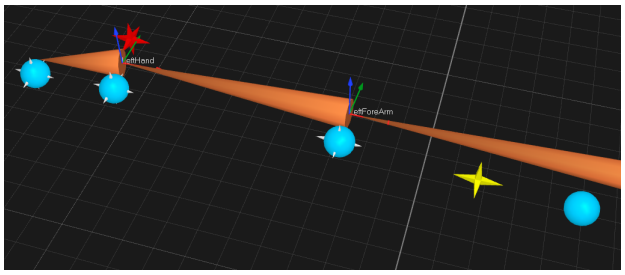
Skeletons are displayed as sets of connected cone-shaped segments. The graphical properties of skeletons can be changed in the **3D view settings**. The visual elements of the skeletons are:

Segments
Cones representing the position and orientation of the rigid parts of the skeleton, with the base indicating the proximal ends and the apex the distal ends.

Segment coordinate system axes
X, Y and Z axis representing the position and orientation of the segments.

Skeleton labels
Text labels attached to skeletons showing their name.

Segment labels
Text labels attached to the respective segments showing their name.



Segment markers
Three dimensional cross-shaped markers indicating the position of the markers

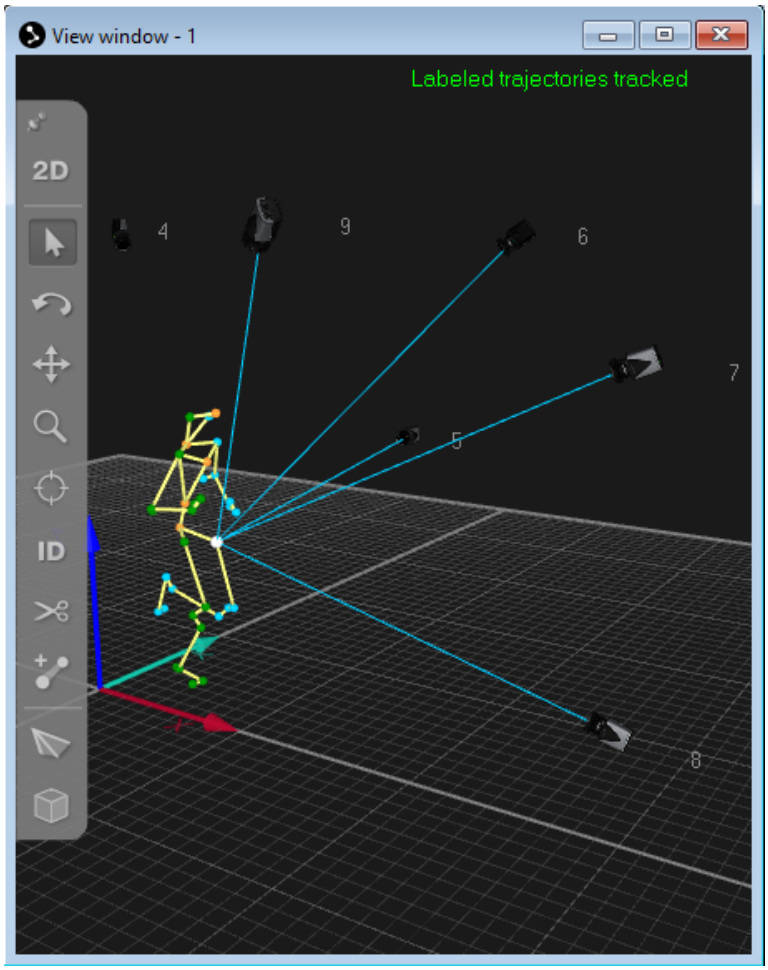
according to the calibrated skeleton definition. The segment markers can have three different colors depending on their status in the current frame.

White: Default color, indicating a good fit with the measured marker position.

Yellow: Indication of a bad fit (deviation of 5 cm or more) with the corresponding measured marker position.

Red: Corresponding marker is missing.

Rays in 3D views



Camera rays in the 3D view window show which cameras have contributed to the selected trajectories. The colors of the rays correspond to the colors of the selected trajectories.

The camera rays are based on a mapping between 2D data of the cameras and the 3D trajectories. Rays are only shown for cameras that have actually contributed to the 3D tracking. The rays may not perfectly intersect with the calculated 3D position of the trajectories since they represent the actual projected 2D position on the sensor based on the used calibration. That means that the rays can be used to visualize the residual of the trajectories.

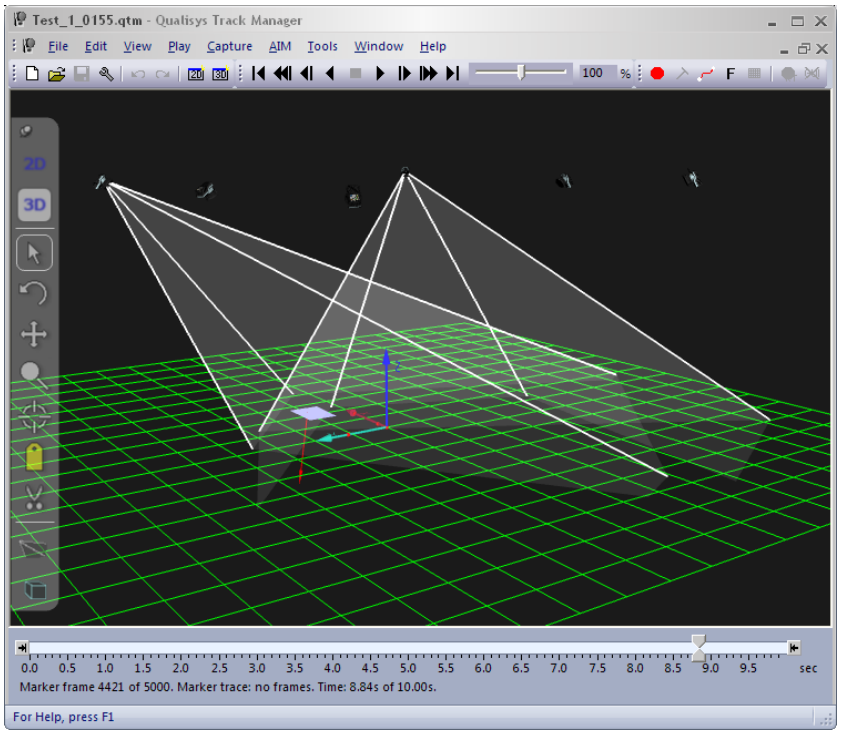
Rays can only be shown if the rays were stored during the processing of the capture. To store the rays, make sure that the **Store** option is enabled in the **Rays** section under **Project Options** > **Processing** > **3D Tracking**, see chapter “Rays” on page 186.

To show rays in the 3D view window, make sure that the **Camera Rays** button on the **GUI Control toolbar** is enabled, or that the option **Enable camera tracking rays** is enabled in the **Rays** section in the **3D view settings**.

Note: Reprocessing of 3D tracking with the ray **Store** option enabled is required to show rays in files captured in QTM versions before QTM 2018.1. Note that by reprocessing the file all manual editing will be lost.

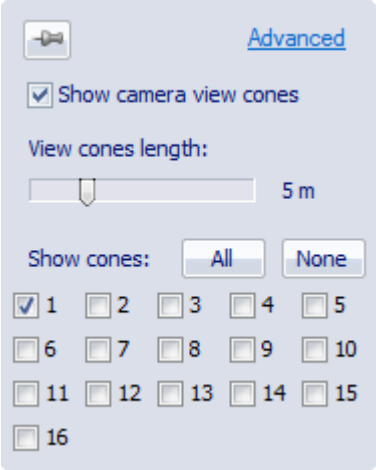
Camera view cones in 3D views

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.

Note: You can also use the **Camera view cones** button on the **GUI Control** toolbar to toggle the display of the cones.

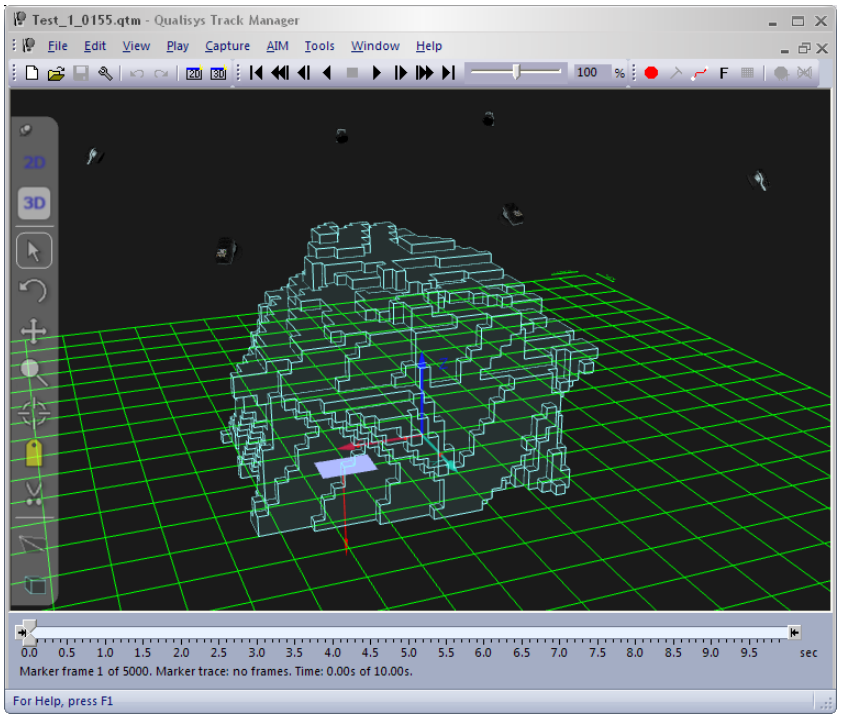


The length of the cones is set in meters by the setting. The cones are by default cut at the floor level, but that can be changed by disabling the **Cut covered volume at floor level** option on the **3D view settings** page in the **Project options** dialog.

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 the previous page.



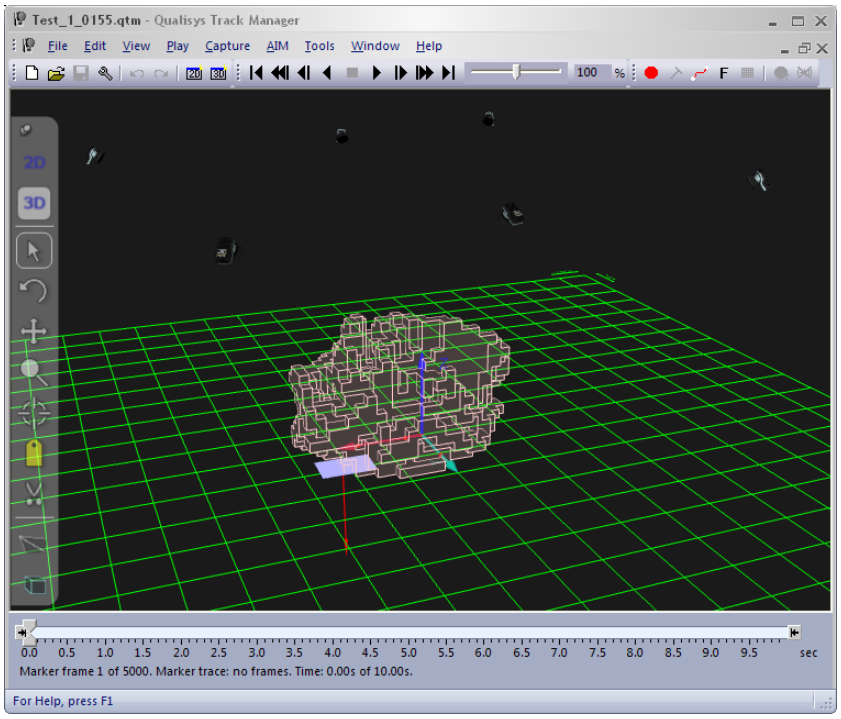
The covered volume is displayed as light blue cubes. It is the volume that is seen by a certain number of cameras, specified by the user with the **Cameras required to consider volume covered** setting on the **3D view settings** page in the **Project options** dialog. The covered volume can be used 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.

Note: You can also use the **Volume** buttons on the **GUI Control** toolbar to toggle the display of the volumes.

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.

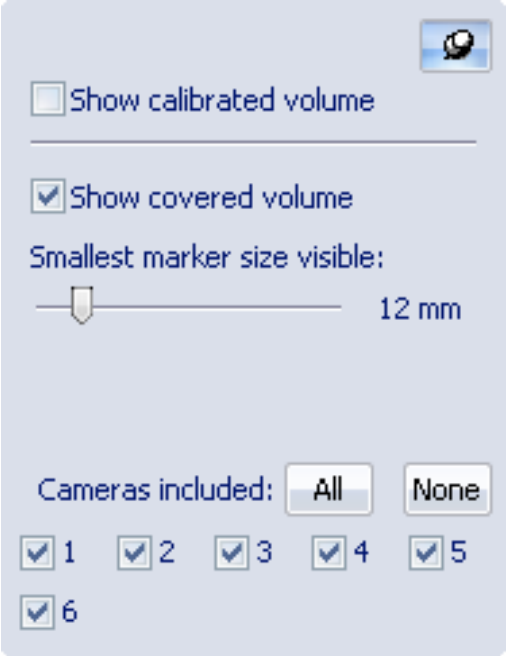
It is important to notice that the covered volume does not consider whether it is likely that markers are occluded by the subject or not. To simulate this you can use the camera selection in the **Volumes** menu see below.



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.

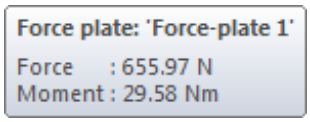
Other objects in 3D views

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.

For the following objects a tooltip with information can be shown when hovering with the mouse on the object.

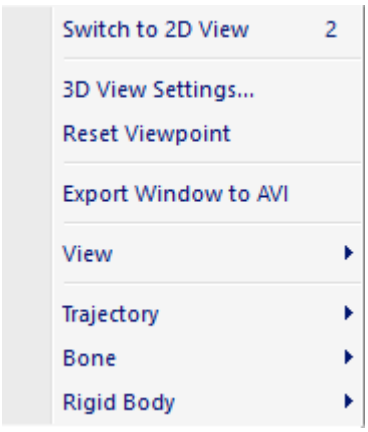
Camera tooltip
Show camera information. The tracking information is the same as in the **File information** dialog, see chapter “3D tracking test” on page 335.

Force data tooltip
Display the current force data when hovering with the mouse on the forece plate or force arrow.



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:

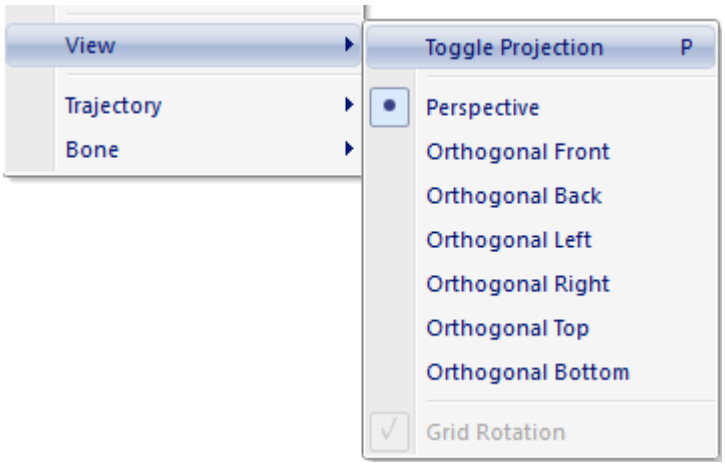
Switch to 2D view
Switch the View window to 2D view.

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, see chapter “3D view settings” on page 244.

Reset viewpoint
Reset the camera viewpoint to center of mass of the tracked markers (same as double clicking in the 3D view window).

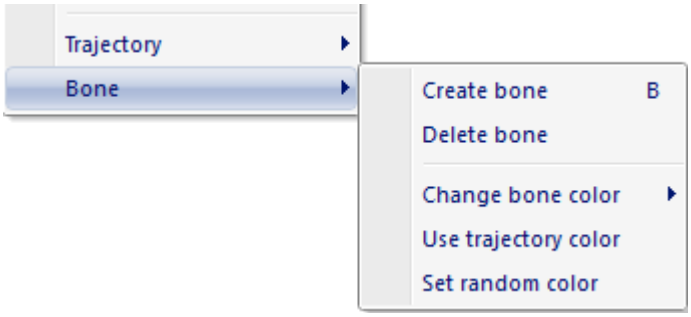
Export window to AVI
Export the current 3D view window to an AVI file, see chapter “Export to AVI file” on page 404.

View
Toggle projection between **Perspective** and **Orthogonal** (keyboard shortcut **P**), and choose viewpoint of orthogonal projection (orthogonal front, back, etc.). When toggling the projection from **Perspective** to **Orthogonal**, the viewpoint snaps to the closest orthogonal projection. The **Grid Rotation** option rotates the grid in accordance with the selected orthogonal view when checked (only available in orthogonal projection mode).



Trajectory
Open the **Trajectory info window** menu for the selected markers, see chapter "Trajectory info window menu" on page 68.

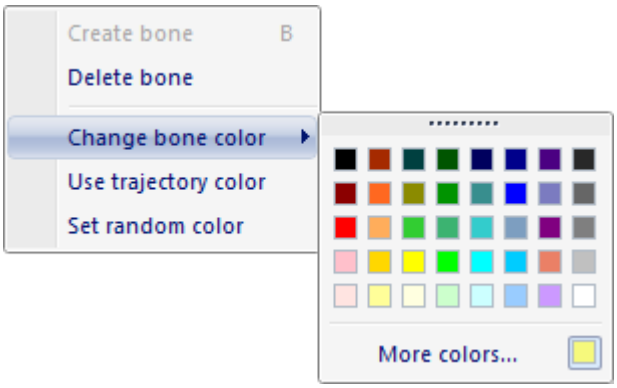
Bone
Create and delete bones and change the bone color. For more information see chapters "Bone menu" below and "Bones in 3D views" on page 49.



Rigid Body
Change mesh settings or change color of selected rigid body. The changes apply to the rigid body definition in the file only.

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.

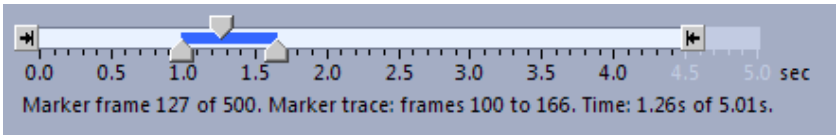
Delete bone
Delete the selected bones.

Change bone color
Select a bone color for the selected bones

Use trajectory color
Use the color of the first trajectory used to create the selected bone.

Set random color
Select a random color for the selected bones.

Timeline control bar



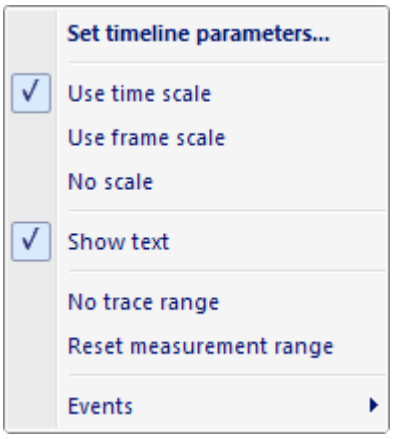
The **Timeline control** bar is shown at the bottom of the QTM window in file mode. It is used to indicate and to select current frame, trace range, measurement range and events.

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, or drag the top slider with the mouse. In the 3D view window you can also use the scrubbing feature (**Ctrl + drag**) to browse through the measurement.

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. You can also use the trace range zooming feature (Shift + Mouse wheel) to increase/decrease the trace range.

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 page 61. For more information about how to use events see chapter “How to use events” on page 383.



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” below. 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.

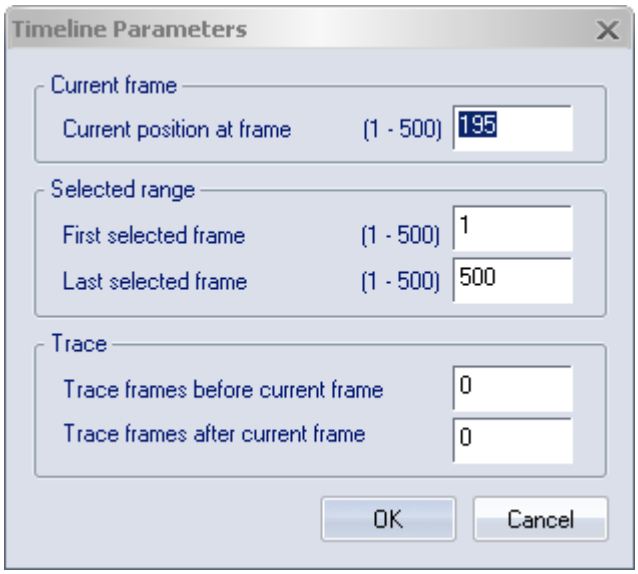
No trace range
Reset the trace range to no trace.

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.

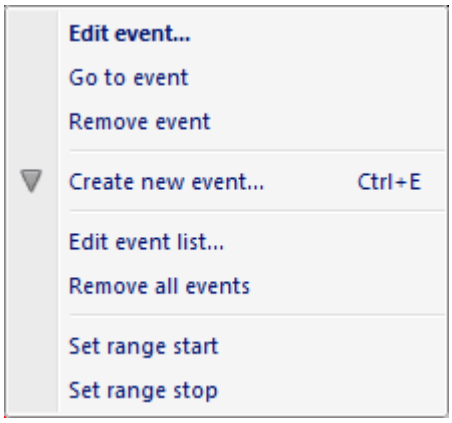
Last selected frame
The last frame in the measurement range, indicated by the scroll box.

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 ones not related to a current event.

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 383.

[Event shortcuts]
The event shortcuts that are listed on the **Events** page in the **Project options** dialog are available on the menu. You can use them to easily create an event with that name and color.

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 384.

Remove all events
Remove all events in the current file.

Set range start
Set the start of the measurement range to the frame of the current event.

Set range stop
Set the stop of the measurement range to the frame of the current event.

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 automatic identification (AIM) is not performed in the 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 68. It is opened by right-clicking on a trajectory or a group of selected trajectories.

Data in Trajectory info windows

Labeled trajectories (35)

Trajectory	Fill Level	Range	Type	X	Y	Z	Residual
L_HEAD	100.0%	1 - 3000	Mixed	410.7	427.7	1571.8	0.30
Part 1	5.1%	1 - 154	Measured	-	-	-	-
Part 2	0.1%	155 - 158	Gap-filled	-	-	-	-
Part 3	94.7%	159 - 3000	Measured	410.7	427.7	1571.8	0.30
R_HEAD	100.0%	1 - 3000	Measured	411.7	265.8	1577.1	0.26
SGL	100.0%	1 - 3000	Measured	513.1	352.0	1592.6	0.33
SME	99.8%	1 - 3000	Measured	-	-	-	-
TV2	100.0%	1 - 3000	Measured	307.2	383.5	1394.9	0.67
TV12	100.0%	1 - 3000	Measured	315.1	393.8	1066.5	0.30

The Trajectory info windows displays the data about each trajectory for the current frame. The data can be sorted by clicking on the column headers, see chapter “Sort trajectories” on page 65.

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.

When a trajectory is empty at the current frame, this is indicated by an open circle, for example .

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 the other trajectory types.

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

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

Edited
A trajectory or part that has been edited using a smoothing function.

Measured slave
A trajectory or part that has been imported from a Twin slave file.

Gap-filled slave
Gap-filled trajectory or part that has been imported from a Twin slave file.

ID#
The sequential id of the short range active marker.

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.

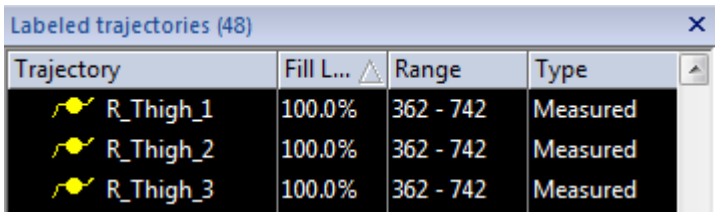
Residual
The average of the different residuals of the 3D point. This is a quality check of the point's measured position.
Note: The residual is not available for 2D tracked trajectories.

Note: The residual is 0 for edited, virtual, gap-filled and interpolated twin slave trajectories.

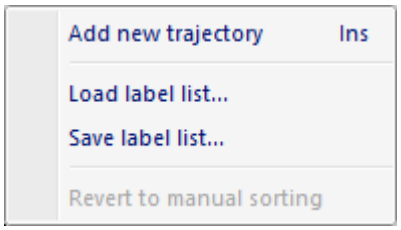
Note: The number after the window name is the number of trajectories in the window.

Sort trajectories

The data in the **Trajectory info** windows can be sorted by clicking the column headers. The trajectories are then sorted in ascending order for that column. Click on the column header to toggle between ascending and descending order.



To revert to the manual sorting in the **Labeled trajectories** window, right-click on the column header to open the **Labels** menu of the window and then select the **Revert to manual sorting** option. The window will then show the trajectories in the order that the file was saved with.



Note: The sorting is only changed temporarily when looking at an open file. The sorting is reverted back to the manual sorting as soon as you close the file. If you want to change the manual sorting of the trajectories in a file you have to use the **Move label up** and **Move label down** options on the **Trajectory info window** menu, see chapter “Trajectory info window menu” on page 68.

Note: If you for some reason cannot revert to manual sorting, then try sorting the trajectories again to activate the **Revert to manual sorting** option.

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.

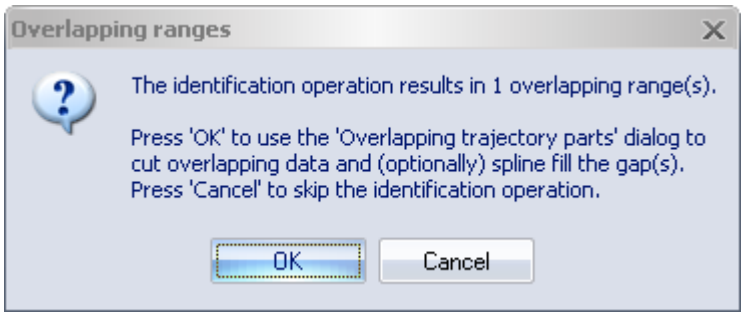
The trajectories can be moved to another **Trajectory info** window by a simple drag and drop, either one by one or in a group. If the trajectory is dropped in the **Labeled trajectories** window the trajectory is labeled 'New 0000', where the number depends on how many trajectories there are in the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

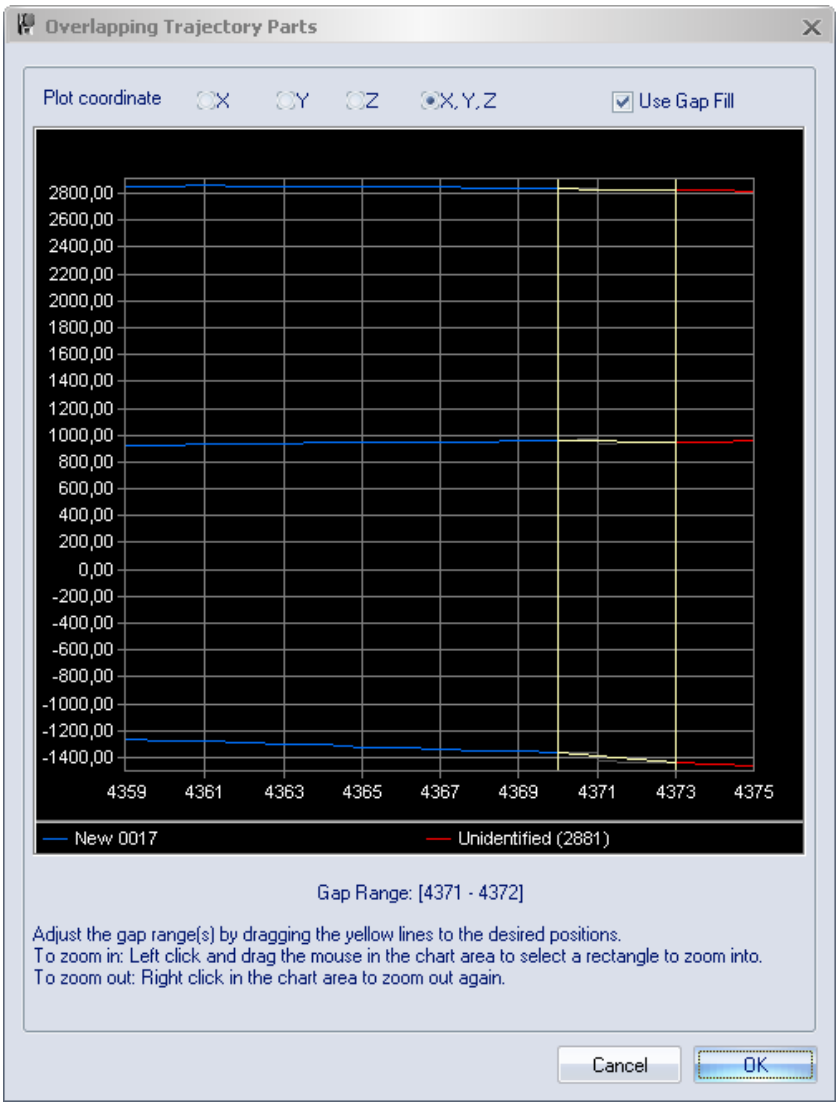
Note: The trajectories can also be drag and dropped from the 3D view window, see chapter “Trajectories in 3D views” on page 48.

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.

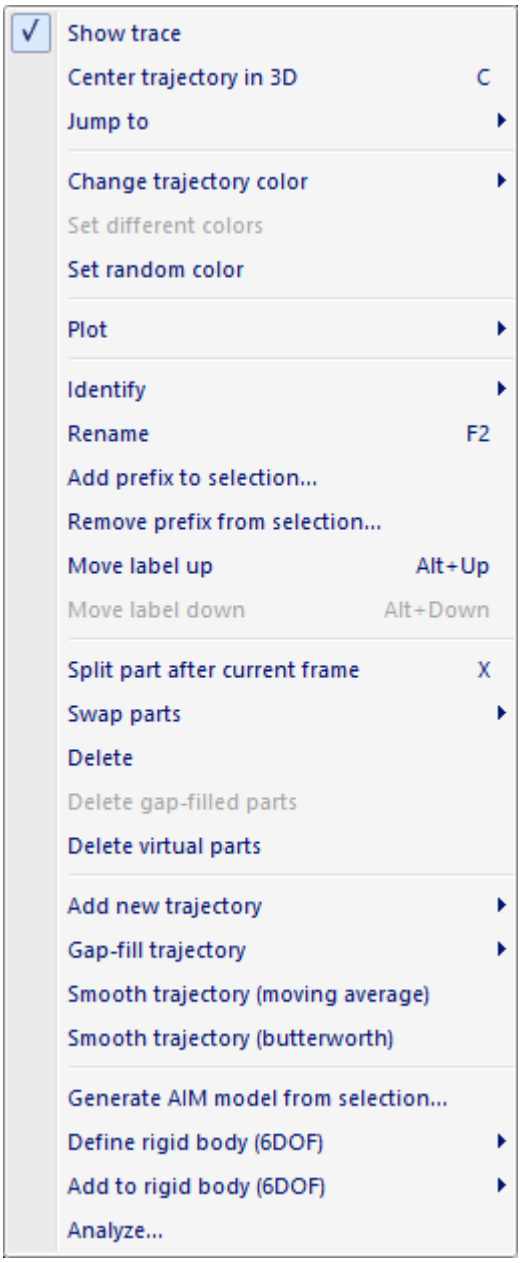
Move the range selectors (the yellow lines) to choose the data that will be included in the joined trajectory. Choose a range where you delete the data that seems to be wrong. It is sometimes better to delete some extra frames and gap fill that part instead of using the original data. The area between 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.
- 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:

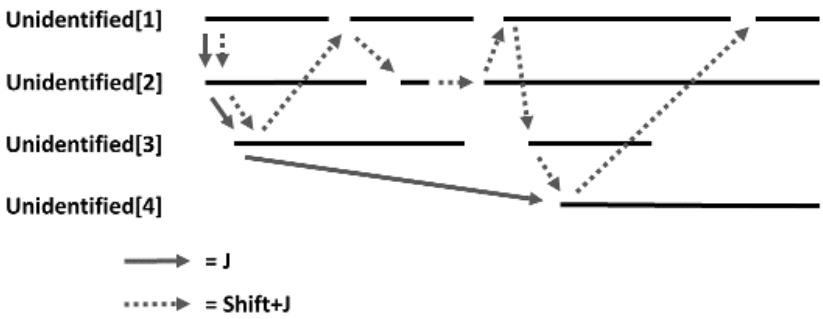
Show trace
Toggle the display of the trace of the selected trajectories.
Note: There are also settings for the display of traces on the **3D view settings** page in **Project options**.

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.

Jump to
Jump in time to the next unidentified trajectory or part (see illustration below). The trajectory or part is selected and the 3D view is centered.

Next unidentified trajectory
Jump to the next unidentified trajectory (keyboard shortcut **J**).

Next unidentified part
Jump to the next unidentified part (keyboard shortcut **Shift+J**)



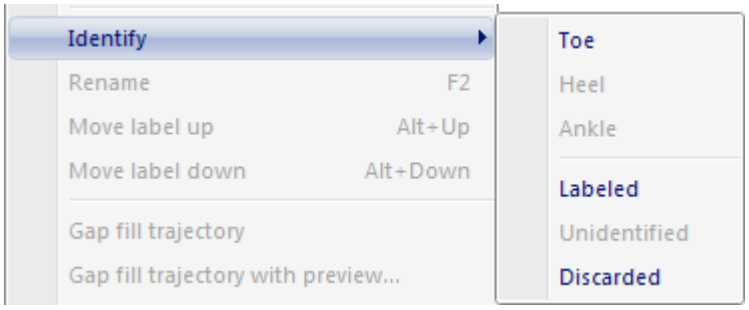
Change trajectory color
Change the color of the selected trajectories.

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 page 72.
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.

Add prefix to selection...
Add a prefix to the selected trajectories.

Remove prefix from selection...
Remove a prefix from the selected trajectories.

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.

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 page 72.

Swap parts
Swap parts between two selected trajectories, see chapter “Swap parts” on page 73.
Note: The parts that are swapped must not overlap any other parts in the trajectories.

Swap all parts
Swap all selected parts in the two trajectories (keyboard shortcut: **S**).

Swap current frame part
Swap the parts of the current frame in the two trajectories (keyboard shortcut: **W**).

Delete
Delete the selected trajectories or parts. If you delete a trajectory in the **Labeled trajectories** window, it is moved to the **Unidentified trajectories** window. More information see chapter “Delete trajectories” on page 72.

Delete gap-filled parts
Delete the gap-filled parts of the selected trajectories or parts.

Delete virtual parts
Delete virtual parts of the selected trajectories or parts.

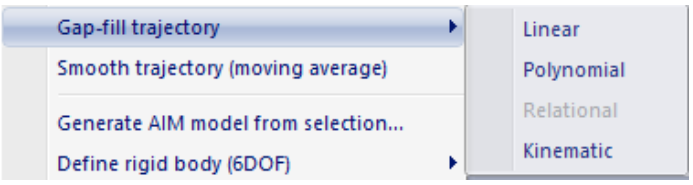
Add new trajectory
Add a new trajectory. The options are:

Empty: Create a new empty trajectory.

Virtual (Average of selected trajectories): Create a new virtual trajectory based on the geometrical average of the currently selected trajectories.

Virtual (Static from current frame): Create one or more new static virtual trajectories at the current frame position of the currently selected trajectory or trajectories.

Gap-fill trajectory
Fill the gaps of the selected trajectories using one of the following gap-fill methods.



Linear
Fill the gaps of the selected trajectories using linear gap filling.

Polynomial
Fill the gaps of the selected trajectories using polynomial gap filling. The maximum gap size is set to the **Max frame gap** setting under **Project Options**, **Trajectories** (default 10 frames) used at the time the file was being processed. For gap filling single trajectories with alternative gap fill settings or methods, see chapter "Filling of gaps" on page 378.

Relational
Apply gap filling of relational type according to the order of the selection. There is no maximum gap limitation. The selection order is as follows:

1. Trajectory to be filled (required)
2. Trajectory used as origin (required)
3. Trajectory used to define the X axis
4. Trajectory used to the define the XY-plane

For more information about relational gap filling, see chapter "Filling of gaps" on page 378.

Kinematic
Fill gaps of selected trajectories using kinematic gap filling based on current skeleton data. The selected trajectories must be included in a skeleton definition.

Smooth trajectory (moving average)
Apply moving average smoothing to selected trajectories. The window size is taken from the **Smooth** settings in the **Settings sidebar** of the **Trajectory Editor** window, see chapter "Settings sidebar" on page 83.

Smooth trajectory (butterworth)
Apply butterworth smoothing to selected trajectories. The cutoff frequency is taken from the **Smooth** settings in the **Settings sidebar** of the **Trajectory Editor** window, see chapter "Settings sidebar" on page 83.

Generate AIM model from selection
Generate an AIM model using the currently selected trajectories only. For more information, see chapter "Generating an AIM model" on page 364.

Define rigid body (6DOF)
Create a new 6DOF body on the **6DOF bodies** page. The available options are:

Current Frame: Create rigid body definition of selected trajectories based on the current frame. This method is automatically applied when in Preview mode.

Average of frames: Create rigid body definition based on the average relative positions of the selected trajectories across the capture. The

advantage of this method is that the statistics of the marker positions are taken into account. The **Bone tolerance** setting of the body will be based on these statistics.

The 6DOF body will be added both to the **6DOF Tracking** page in **Project Options** and in the current file. The origin of the rigid body will be the geometric average of the included marker positions. The orientation corresponds to the orientation of the rigid body at the first frame of the 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.

Add to rigid body (6DOF)
Add current frame positions of selected markers to a rigid body. The point(s) will be added to the rigid body definition in the current file and to that in the **Project Options** if it exists.

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 94.

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.

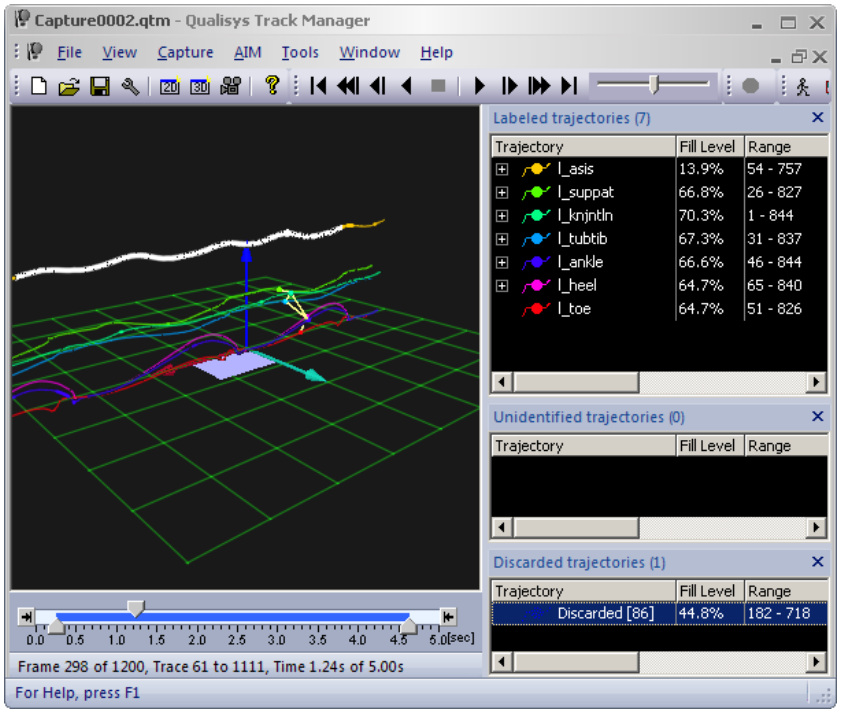
You can also use the **Cut trajectory trace** tool in the 3D view window to split a trajectory. Click on a trace with the tool to split the trajectory at that position.

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**, the trajectory is moved to the **Unidentified trajectories** window. Then if you delete it in **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.



Swap parts

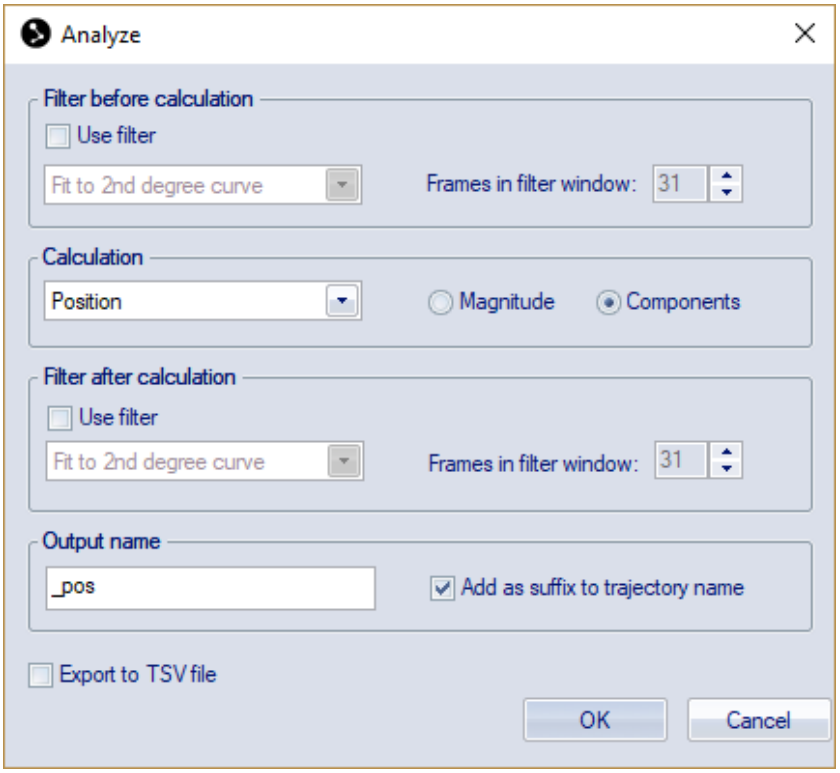
The **Swap parts** function on the **Trajectory info window** menu makes it easy to swap parts that have been added to the wrong trajectories. Follow these steps to the swap parts.

1. Expand the trajectories in which you want to swap parts, by clicking on the + sign next to the label.
2. In the 3D view check where the tracking error starts and stops.
   a. Make sure to check that there is a new part and the start and stop. Otherwise you need to split that part for the swap to work.
3. Select the parts that needs to be swapped and use the **Swap parts**/**Swap all parts** option on the **Trajectory info window** menu, the shortcut is s.
   a. If it is just one part in each trajectory that needs to be swapped you can use the **Swap current frame part** option instead. This option will automatically swap the parts of the current frame.
   b. Parts can be selected directly in the 3D view window, by holding down Alt+Shift while clicking on a trace.
   c. Make sure that you delete any gap filled parts at the start and end of the swap.

Note: If there is any overlap with parts outside the parts that are swapped, then QTM will give you a warning and you may select to swap all of the parts that are overlapped.

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. To export the results to TSV file, check the **Export to TSV file** option.



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.

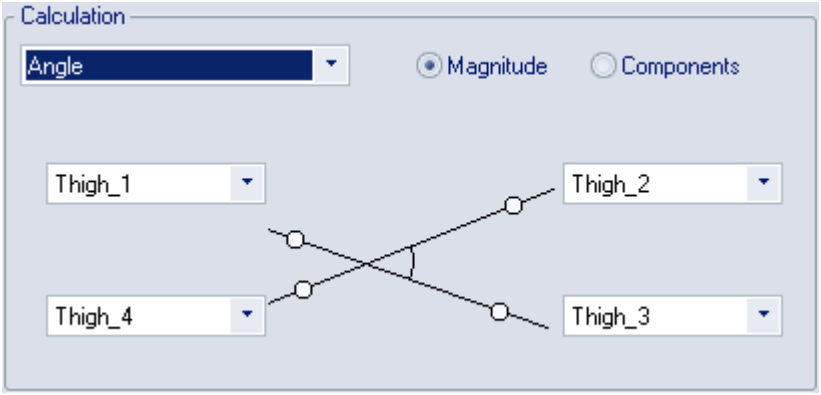
Acceleration
Calculates the acceleration of the trajectories.

Distance
Calculates the distance between two trajectories over time.
Note: Distance can only be calculated when just two trajectories are selected.

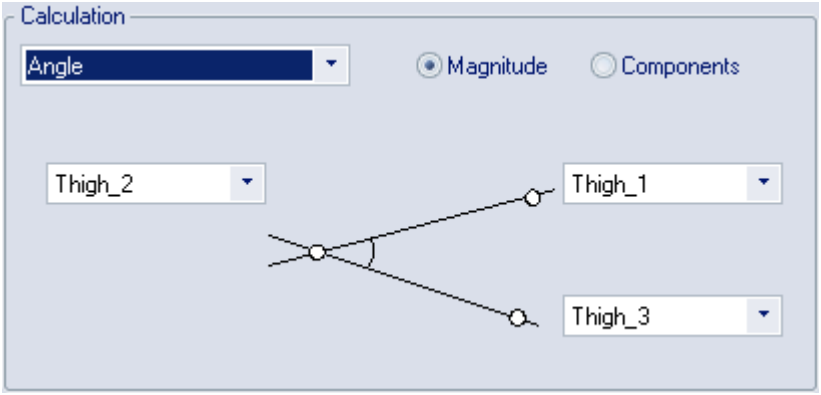
Distance traveled
Calculate the distance that the trajectory has traveled in the measurement. The distance will increase with every frame that marker position has changed more than 0.2 mm compared with the last time the marker moved. The 0.2 mm hysteresis is there to remove the static noise, otherwise 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.

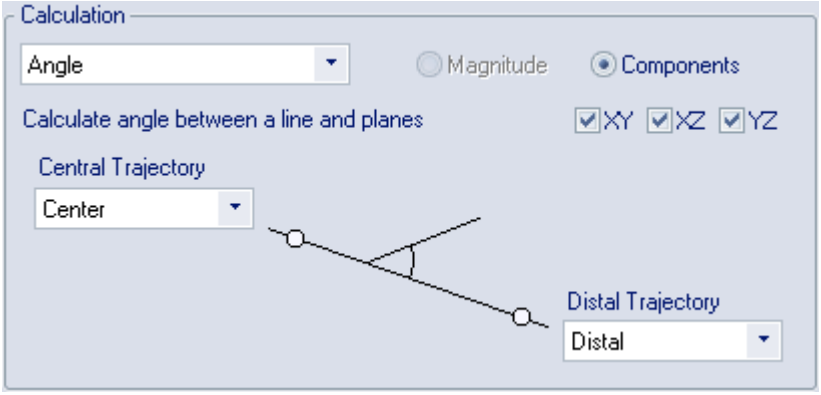
Angle
Calculates the angle between two lines in space, using either two, three or four trajectories. Select the trajectories that you want to match with the markers in the diagram to calculate the angle between the lines. For three and four markers the angles are calculated in the range 0 – 180 degrees. 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 acceleration 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.

Label lists

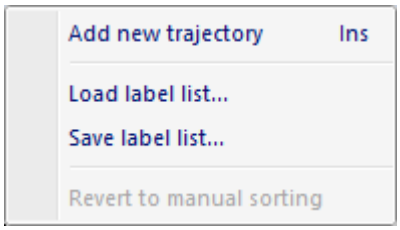
The labels of the trajectories in the **Labeled trajectories** window can be saved in label lists. The label lists are controlled from the **Labels** menu.

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 trajectory
Add a new empty trajectory 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.

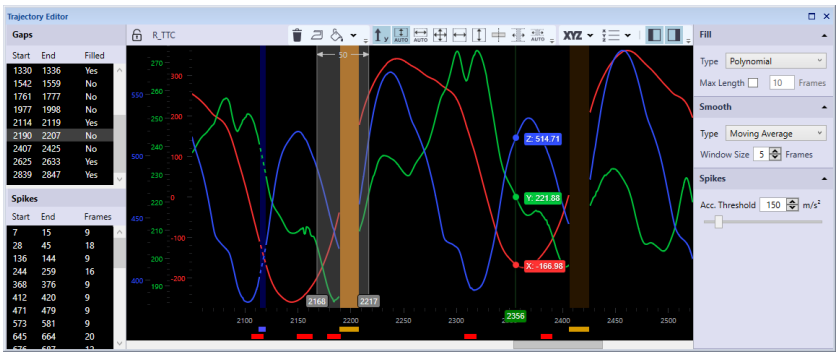
Revert to manual sorting
Revert the sorting of the **Labeled trajectories** window to the sorting that the file was saved with.

Trajectory Editor window

The **Trajectory Editor window** can be used for display and editing of trajectory data. The editing facilities include several methods for gap filling and smoothing of data.

For information on how to use the **Trajectory Editor**, see chapter “Editing of trajectories” on page 377.

The Trajectory Editor window can be docked into QTM or it can be used as a floating window. For docking or floating, click on the title bar of the window and drag it around. By dragging the window onto the smart docking markers you can dock it at the desired position in QTM.



The **Trajectory Editor window** contains the following elements:

Plot area, see chapter “Plot area” below.

Trajectory Editor toolbar, see chapter “Trajectory Editor toolbar” on the next page

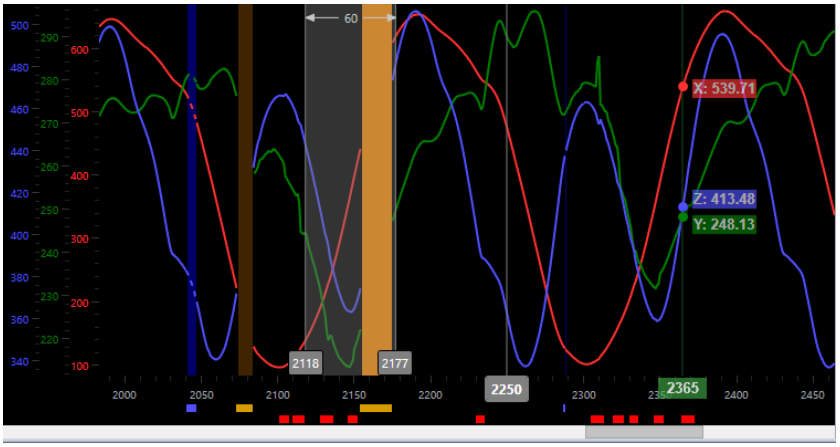
Points of Interest Sidebar, see chapter “Points of Interest sidebar” on page 82.

Settings Sidebar, see chapter “Settings sidebar” on page 83.

Trajectory Editor window menu, see chapter “Trajectory Editor window menu” on page 84.

Keyboard shortcuts, see chapter “Trajectory Editor shortcuts” on page 118.

Plot area



The plot area of the **Trajectory Editor** window shows the selected data series. The lay out of the plot area depends on the view settings. The plot area contains the following elements.

Data series
Red, green and blue lines for X, Y and Z, respectively.

Vertical axis (optional)
Values of respective data series. The units are mm, m/s and m/s^2 for position, velocity and acceleration, respectively.

Time axis (frames or time in seconds)
Horizontal axis with frame or time values.

Tool tip
Show data value at current mouse pointer position

Current frame
Gray line and frame/time value

Selection
Gray area, edge values and width

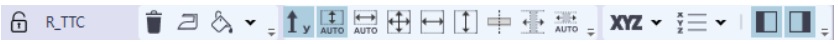
Gap indicators
Amber and blue indicators below the time axis indicating unfilled and filled gaps, respectively, and corresponding areas in the plot area.

Spike indicators
Red indicators below the time axis indicating detected spikes.

The view can be modified via the buttons on the toolbar, see chapter “Trajectory Editor toolbar” below.

For an overview of keyboard shortcuts and mouse gestures, see chapter “Trajectory Editor shortcuts” on page 118.

Trajectory Editor toolbar



The toolbar of the **Trajectory Editor** window contains the following elements.

The left side of the toolbar shows the label of the currently selected trajectory. The lock button to the left of the label can be used to lock the currently selected trajectory.

To the right of the label the action buttons are located. The action applies to the current selection of the trajectory. The label of the selected trajectory is shown on the right side of the action buttons. If needed, the latest actions can be undone by pressing **Undo** in the **Edit** menu or **Ctrl** + **Z**.

Delete (Del)
Delete current selection. The selection will be permanently deleted.

Smooth (S)
Smooth current selection of trajectory using the smoothing method specified in the Trajectory Editor settings sidebar.

Fill (F)
Fill gaps within current selection using the fill method specified in the Trajectory Editor settings sidebar. When expanding the button, you can choose to fill all gaps (**F**) or only unfilled gaps (**Shift** + **F**).

The remaining buttons can be used to modify the view of the data series and the lay out of the Trajectory editor window.

Vertical Axis
Toggle visibility of vertical axes.

Auto Zoom Vertical (A)
Toggle zoom vertical axis automatically.

Auto Zoom Horizontal (B)
Toggle zoom horizontal axis automatically when selecting trajectories.

Zoom To Extents (X)
Zoom horizontal and vertical axis to data extents.

Zoom Horizontal (H)
Zoom horizontal axis to data extents

Zoom Vertical (V)
Zoom vertical axis to data extents

Center Current Frame (C)
Center visual range on current frame

Zoom To Selection (Z)
Zoom to currently selected range

Zoom to Point of Interest(G)
Toggle automatic zoom to selected point of interest.

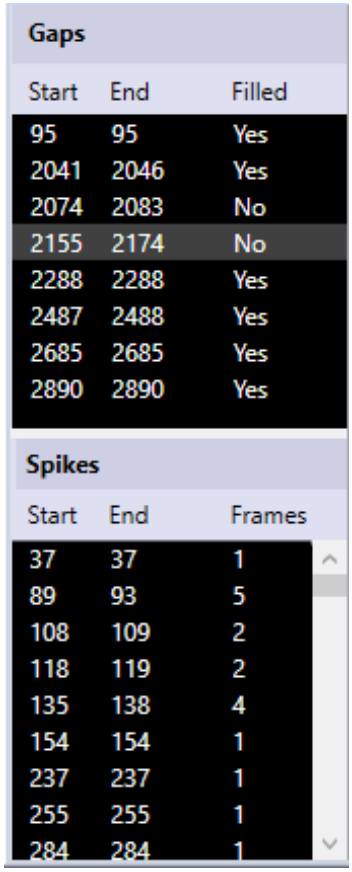
Series
Select data series to show in the chart (X, Y, Z, (R)esidual, (V)elocity, (A)cceleration)

View
The options are: (1) Combined view, (2) Component view; (3) Merged view.

Points of Interest Sidebar
Show/hide Points of Interest Sidebar.

Settings Sidebar
Show/hide Settings Sidebar.

Points of Interest sidebar



Gaps

Start	End	Filled
95	95	Yes
2041	2046	Yes
2074	2083	No
2155	2174	No
2288	2288	Yes
2487	2488	Yes
2685	2685	Yes
2890	2890	Yes

Spikes

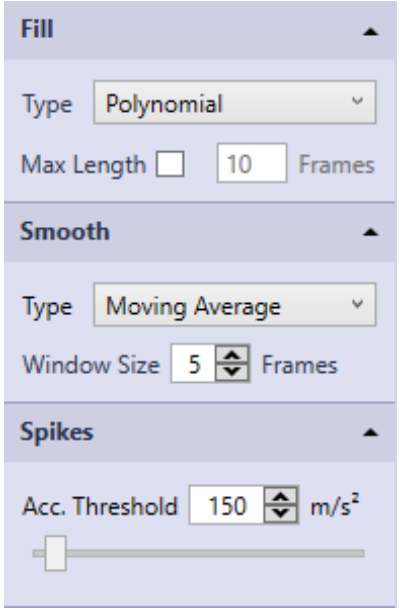
Start	End	Frames
37	37	1
89	93	5
108	109	2
118	119	2
135	138	4
154	154	1
237	237	1
255	255	1
284	284	1

The **Points of Interest sidebar** contains information about gaps and detected spikes.

The **Gaps** pane shows the start and end frames of gaps contained in the selected trajectory as well as their fill status.

The **Spikes** pane shows the start and end frames of detected spikes, as well as their width (number of frames). The detection is based on the acceleration level, which can be set in the **Trajectory Editor settings** (see chapter “Settings sidebar” on the facing page).

Settings sidebar



In the settings sidebar of the **Trajectory Editor** window you can select methods for gap filling and smoothing, and change their parameters. In addition, you can set the spikes detection level.

The following gap fill types are available. For more information on how to use the different gap fill types, see chapter “Filling of gaps” on page 378.

Static
Gap filling by adding a fixed 3D position (virtual point).

Linear
Gap filling by means of linear interpolation.

Polynomial (default)
Gap filling by means of a cubic polynomial interpolation.

Relational
Gap filling by interpolation based on the movement of surrounding markers selected by the user.

Virtual
Gap filling by adding a virtual trajectory based on the movement of surrounding markers selected by the user and an optional offset.

Kinematic
Kinematic gap fill of markers associated with skeleton segments or rigid bodies.

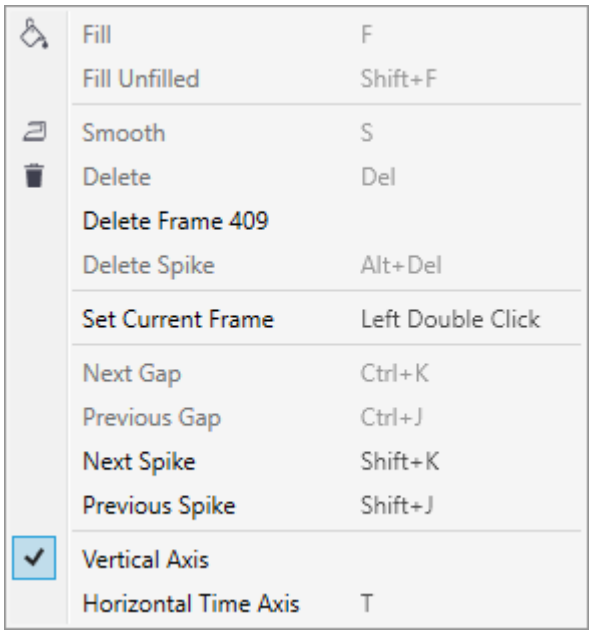
The following smoothing methods are available. For more detailed information about smoothing, see chapter “Smoothing” on page 381.

Moving average
Smoothing by means of an unweighted moving average.

Butterworth
Smoothing by means of a Butterworth filter.

The spikes detection is based on a threshold of the acceleration magnitude. The threshold level can be adjusted to control the strictness of the spike detection.

Trajectory Editor window menu



The **Trajectory Editor window** menu contains options for the selected trajectory. The menu can be accessed by right-clicking in the plot area of the **Trajectory Editor**.

The following options are available:

Fill
Fill all gaps included in the selected frame range using the current gap fill settings.

Fill Unfilled
Fill all unfilled gaps included in the selected frame range using the current gap fill settings.

Smooth
Smooth the data included in the selected frame range using the current smoothing settings.

Delete
Delete the data included in the selected frame range.

Delete Frame xxx
Delete the frame at the mouse pointer position.

Delete Spike
Delete spikes included in current selection. Only the frames with acceleration values above the current acceleration threshold will be deleted, excluding the additional margins.

Set Current Frame
Set current frame to mouse pointer position.

Next Gap
Select next gap.

Previous gap
Select previous gap.

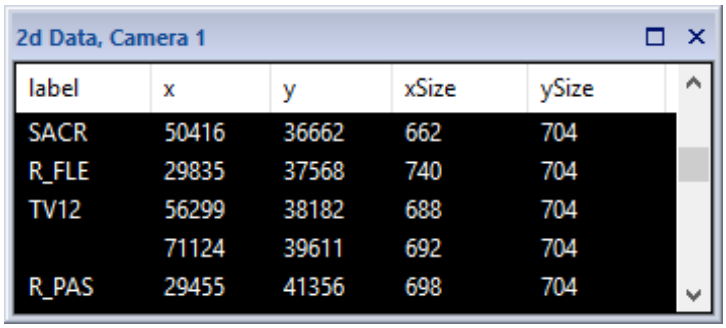
Next Spike
Select next spike.

Previous Spike
Select previous spike.

Vertical Axis
Show/hide vertical axis information.

Horizontal Time Axis
Toggle between frame or time (seconds) units for the horizontal axis.

Data info window

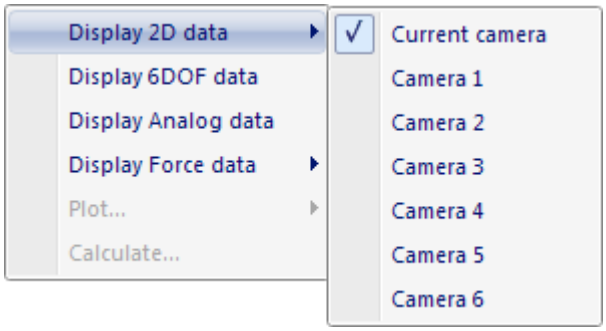


The data for the current frame can be viewed in the **Data info** windows. There can be three **Data info** windows open at the same time, so that different types of data can be displayed. Open the windows by clicking **Data info 1** to **Data info 3** on the **View** menu. Use Ctrl+D to toggle the display of the Data info 1.

The **Data info** windows can be opened both in file mode and in preview mode. It is a tool window that can be placed anywhere and is by default placed on the left side of the main window.

All of the data (except 3D 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 the 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.

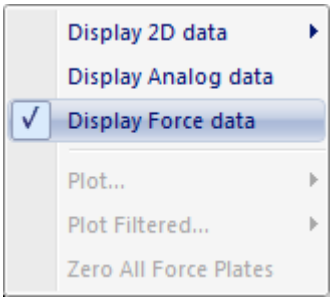
Display Force data
Choose which force-plate's force data that will be displayed.

Plot
Plot the selected data, see chapter "Plot window" on page 94.

Plot Filtered
Filtered plot for the selected data, see chapter "Plot window" on page 94.

Calculate
Calculate magnitude of distance from origin of a 6DOF body, see chapter "Calculate" on page 93.

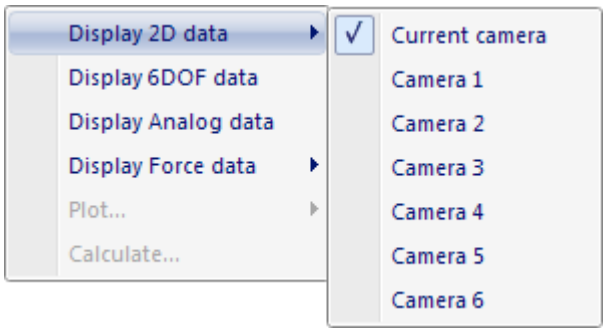
Zero All Force Plates (only when displaying force data)
While in preview zero all of the force plates, used for Kistler and digital AMTI plates.



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.

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 1

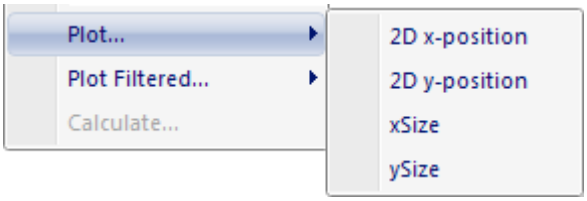
label	x	y	xSize	ySize
SACR	50416	36662	662	704
R_FLE	29835	37568	740	704
TV12	56299	38182	688	704
	71124	39611	692	704
R_PAS	29455	41356	698	704

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:

label
The labeled 3D trajectory (current frame) associated with the marker. The label can only be displayed if the option to store rays is checked in the on the **3D Tracking** page under **Project Options**.

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

xSize, ySize
The size of the marker in subpixels.



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 94.

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 202.

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.

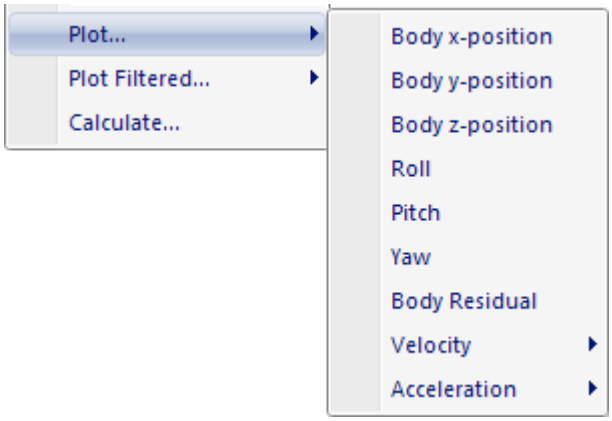
Note: The rotation is positive when the axis rotates clockwise when looking in the positive direction of the axis and it is applied in the order: roll, pitch and yaw. E.g. to mimic the rotation of a certain rigid body in a certain frame follow these steps:

- Align the local coordinate system with the global.
- Rotate roll to the correct angle.
- Rotate pitch to the correct angle. Remember that the Y-axis has moved because of the roll.
- Rotate yaw to the correct angle. Remember that the Z-axis has moved because of roll and pitch.

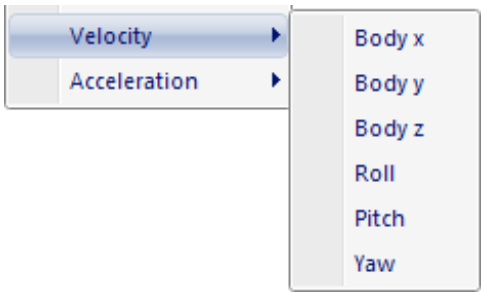
Note: If you change the Euler angle definition the example above will not be correct and the name of the angles can be changed.

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 94.

Skeleton data information

Skeleton Data (Local coordinates)

Skeleton	Segment	X	Y	Z	Roll	Pitch	Yaw
VF	Hips	504.14	247.46	970.30	1.81	-0.07	4.65
VF	Spine	0.33	0.84	127.40	-0.12	0.33	-0.42
VF	Spine1	-0.00	10.43	118.28	0.03	-0.32	-0.46
VF	Spine2	-0.00	15.65	153.76	0.28	0.36	-0.10
VF	LeftShoulder	-26.08	37.32	119.56	-1.77	-0.11	-1.57
VF	LeftArm	-153.20	-0.00	3.95	-1.38	-0.96	-3.75
VF	LeftForeArm	-294.29	0.00	-0.00	-1.00	-0.16	0.01
VF	LeftHand	-243.44	0.00	-0.00	-0.63	0.55	2.90
VF	RightShoulder	29.09	35.71	119.63	-1.11	-0.93	-0.92
VF	RightArm	122.78	0.00	-3.95	-0.19	-3.85	2.07
VF	RightForeArm	302.02	0.00	-0.00	-0.40	-0.27	0.14
VF	RightHand	242.47	-0.00	0.00	0.20	1.63	-3.70
VF	Neck	-0.00	15.65	153.76	-0.45	0.32	-0.24
VF	Head	7.11	36.63	107.71	-1.51	2.06	-0.76
VF	LeftUpLeg	-84.09	43.04	-14.51	0.82	3.50	-1.69
VF	LeftLeg	0.00	-3.76	-431.16	-5.55	-5.84	1.28
VF	LeftFoot	-0.00	-22.94	-437.68	3.23	2.07	-0.55
VF	LeftToeBase	0.00	115.37	-62.75	0.10	0.90	-0.23
VF	RightUpLeg	84.09	43.04	-14.51	-1.21	-2.63	-7.00
VF	RightLeg	0.00	-3.74	-428.47	-3.58	3.62	-2.38
VF	RightFoot	-0.00	-23.62	-450.63	2.76	-0.83	-1.29
VF	RightToeBase	0.00	114.67	-64.56	-0.11	-0.35	0.11

The data of the skeletons in the current frame can be viewed in the **Data info** window. The skeletons will be shown in the same order as on the **Project Options > Skeleton solver** page and the data will use the definitions for angles and local coordinate system on the **Project Options > Euler angles** page.

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

Skeleton
Name of the skeleton.

Segment
Name of the skeleton segment.

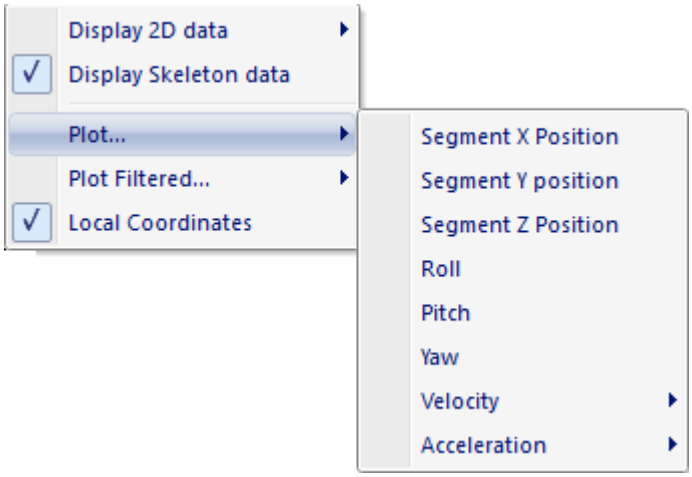
X, Y, Z
Segment 3D positions.

Roll, Pitch, Yaw
Segment orientation expressed in Euler angles. For custom Euler angle definitions the order and names of these columns are as defined by the user.

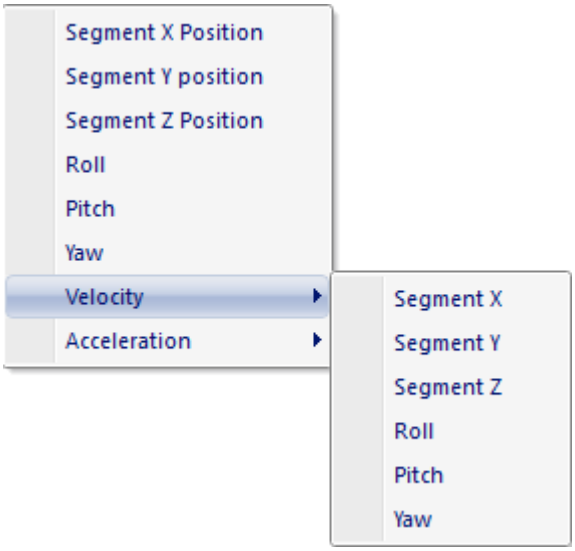
The following menu options are available when right clicking in the **Skeleton data information window**.

Local Coordinates
Display local coordinates of segments relative to their respective parent segment when checked. Display global coordinates of segments when unchecked.

To plot skeleton segment data, select one or more segments, click **Plot** or **Plot filtered** in the **Data info window** menu and select the type of data to plot. With **Plot filtered** you can apply a **Fit to 2nd degree curve** or **Moving average** filter.

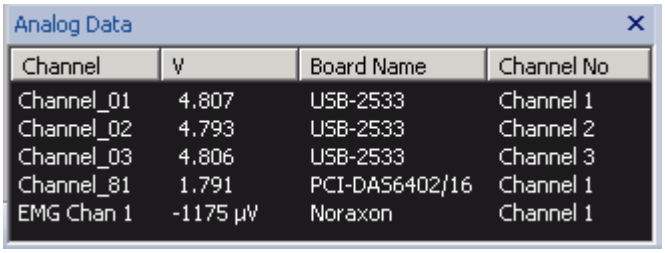


You can also plot the velocity and acceleration in the three directions of the coordinate system for segment data and the angular velocity and acceleration for the three rotation angles. It is recommended to use **Plot filtered** and apply the filter before the calculation for **Velocity** and **Acceleration**,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



For information about the **Plot** window see chapter“Plot window” on page 94.

Analog data information



Analog Data

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.

Value
The voltage input to the analog board.
Note: If the data is not in V then the unit is displayed after the data. If you have a sensor on the EMG that gives you other data than V then it is in the SI unit of that type of data, except for accelerometers that are in g.

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 94.

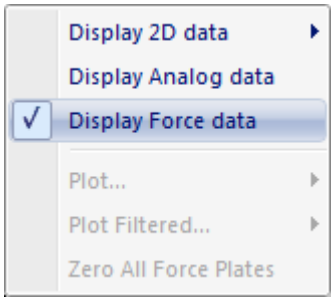
Note: The number of seconds in the analog RT plot is specified on the **GUI** page in the **Project options** dialog, see chapter “GUI” on page 241.

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 168.
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.

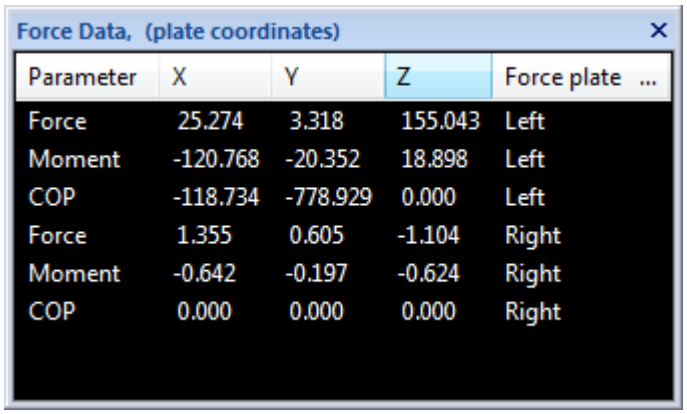
Apply drift compensation
Activate/deactivate this option to apply/remove drift compensation for the selected channels, i.e. where the line is blue. The option can only be changed in a file and settings used will be those set before the measurement on the **Analog board** page in the **Project options** dialog, see chapter“Compensate for analog offset and drift” on page 168.
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, to show the force data for all force plates in the **Data info** window.

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 386.



Parameter	X	Y	Z	Force plate ...
Force	25.274	3.318	155.043	Left
Moment	-120.768	-20.352	18.898	Left
COP	-118.734	-778.929	0.000	Left
Force	1.355	0.605	-1.104	Right
Moment	-0.642	-0.197	-0.624	Right
COP	0.000	0.000	0.000	Right

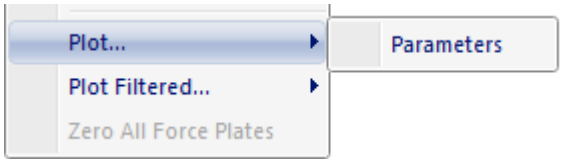
The **Data info** window will then contain data for the force plate's **Force**, **Moment** and **COP** (Center Of Pressure) vectors. The data is for the actual force on the force-plate. The displayed data is for the current frame in the following five columns:

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, if you are using the **Local (Force plate)** setting. The Z direction of the force plate coordinate system is pointing down.

Force plate
The name of the force plate for the data in that row.

The coordinate system that is used is displayed in the title of the window, **plate** or **lab**. You can switch between the coordinate system on the **Force data** page in **Project options** dialog, see chapter "General settings" on page 210.

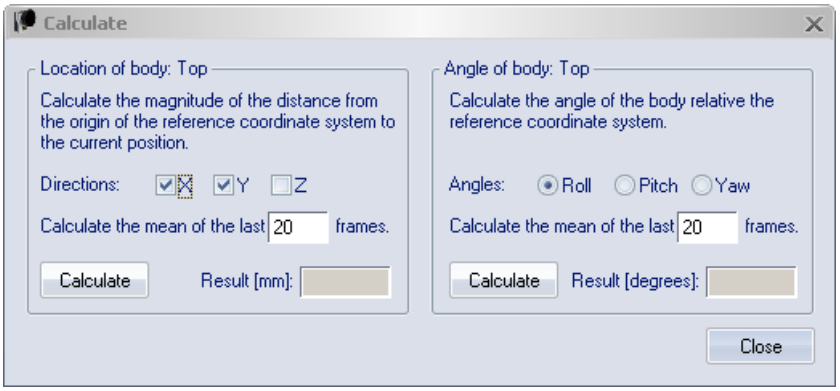


Note: The **Force** and the **COP** is also visualized with a vector in the **3D view** window. The vector is the opposite force applied to the force plate, i.e. opposite to that shown in the **Data info** window.

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 next 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 241.

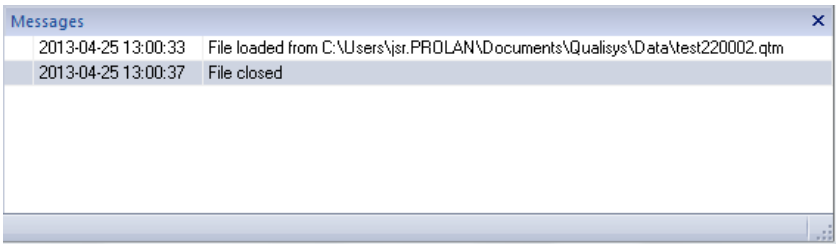
Calculate



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.

Messages window



The **Messages** window contains the processing messages, camera error messages and some status information about what has happened in QTM, since it was last started. The messages are displayed with the oldest first in the list, scroll down to the bottom to see the latest messages.

The following mouse actions are supported:

Use the **scroll wheel** to scroll up and down the message list.

Double click on an error to open an error dialog with more specific information.

Right click and select **Clear list** to clear the message list.

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 names as the rows in the **Data info** window. If the graph contains 3D data of the trajectories, the colors of the series are the same as those of the trajectories.

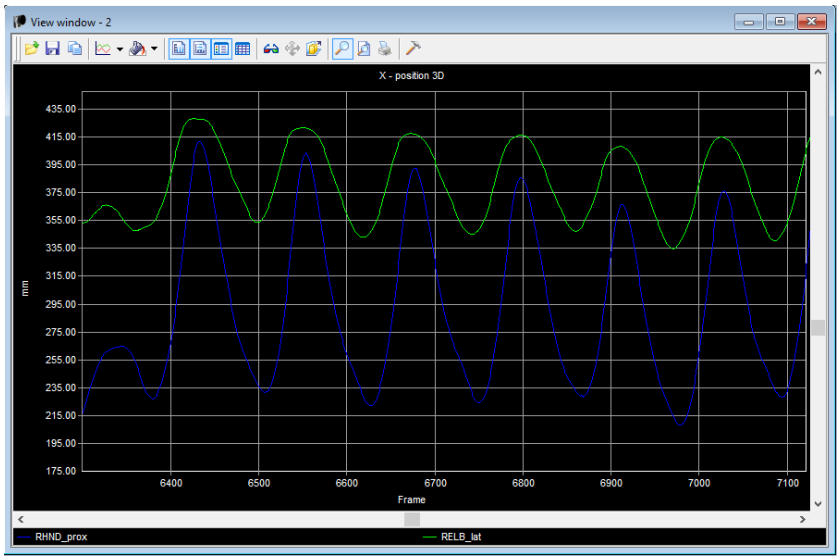
Note: If the measurement range is smaller than the initial range, the parts of the trajectory which are outside the measurement range are grey.

The **Plot toolbar** shows the available plot options. For more detailed information, see chapter “Toolbar” on the facing page.

Note: currently, it is not possible to access plot properties.

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.



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.

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

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. 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.

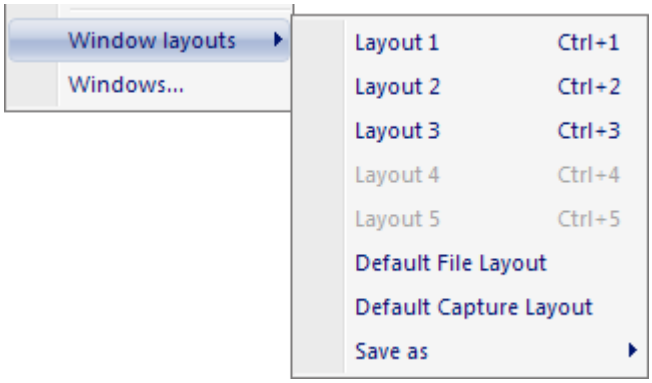
The window handling is done either manually with the mouse or with the commands in the **Window** menu. Manually you can move and resize the window to the desired place and size. While the following actions can be performed from the **Windows** menu:

- 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 the facing page.

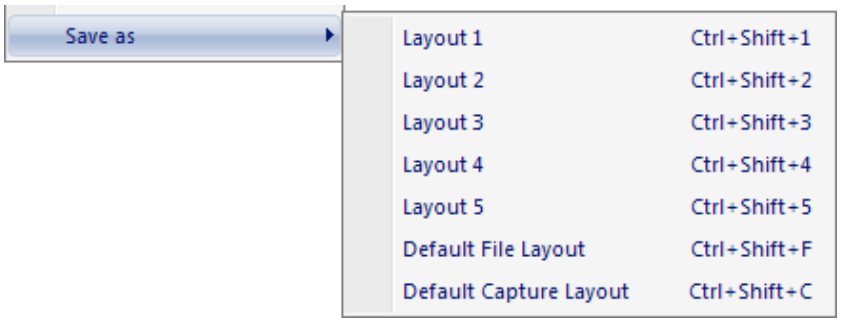
Window layout

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.

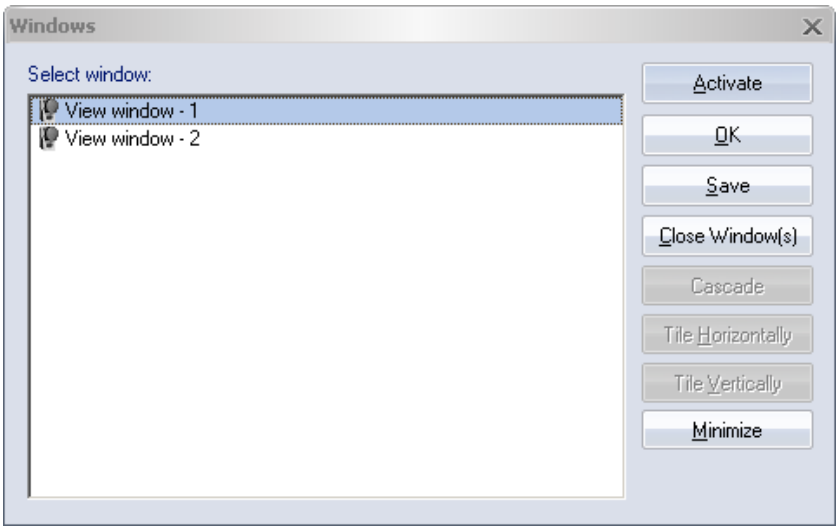
Trajectory info windows

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. The following file types can be opened in QTM:

QTM files: capture files (.qtm) and calibration files (.qca).

C3D files: import of C3D files (.c3d), see chapter "C3D import" on the next page.

Batch Process...
Open configuration dialog for the batch processing function, see chapter "Batch processing" on page 329.

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.
Note: When saving files you can always go to the **Project data folder** with the link at the top of the list to the left in the **Save** dialog.

Close
Close the active capture file.

New project...
Create a new project, see chapter "Creating a new project" on page 17.

Open project...
Browse for an existing project.

Save project
Save the current project settings to the *Settings.qtmproj* file.

Rename project
Rename the current project, this will rename the project folder.

Open project folder
Open the project folder where for example the calibration files are saved, see chapter "Project folder" on page 23.

Manage projects...
Open the Manage Projects dialog, see chapter "Manage projects" on page 18.

Settings management

Backup and Restore
Open a dialog to save and restore backups of the project settings, see chapter "Backup of project settings" on page 22.

Import from another project
Browse for the project that you want to import the settings from. All of the settings in your current project will be overwritten.

Project presets
Open a dialog to save project presets that can be used when creating projects, see chapter “Project presets” on page 22.

Export
Export the capture file to one of the following formats, see chapter “Data export to other applications” on page 389.

To TSV

To C3D

To MAT

To AVI

To FBX

Import

Add link to video file
Import a link to an AVI file, see chapter “Import video link” on page 474.

Recent projects
List of recently opened projects

1-10...
The last ten opened QTM files.

Exit
Exit the application.

C3D import

When opening a C3D file, it will be imported in QTM. The import includes:

- 3D data
- Analog data
- Force data
- Force plate positions

You can display, manage and edit the imported data using the regular QTM functionality. The imported data can be saved as a QTM file.

Note: It is not possible to retrack the trajectories since the imported file does not contain any 2D data.

Note: When importing force data, a force plate may be converted to a generic type-2, type-3 or type-4 plate. The generic types are used when there is not enough information to select one of the vendor specific force plates.

Note: When exporting the imported file as C3D, the exported C3D file may have a different format than the original C3D file.

Note: Not all C3D files may be compatible with the import function. Contact Qualisys support at support@qualisys.com if you have problems to import your C3D files.

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.

Trim Measurement
Trim the measurement to the current selected range on the time line.

Note: When trimming the file, the time stamp of the file corresponding to the start of the capture is recalculated.

Find
Search for the first occurrence of the search term in the **Project data tree**, see chapter “Project data tree” on page 19.

Find Next
Search for the next occurrence of the search term in the **Project data tree**.

Trajectory
The **Trajectory info window** menu for the selected trajectories, see chapter “Trajectory info window menu” on page 68.

Bone
The **Bone** menu for creating and deleting bones, see chapter “Bone menu” on page 58.

Labeled trajectories
The **Labeled trajectories** menu for the trajectories in the **Labeled trajectories** window, see chapter “Labels menu” on page 77.

Events
The **Events** menu for creating and editing events, see chapter “Events menu” on page 61.

View

Trajectory info windows
For information about **Trajectory info** windows, see chapter “Trajectory info windows” on page 63.

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 chapter “Toolbars” on page 109.

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.

GUI Control toolbar
Toggle the display of the **GUI Control** toolbar.

Trajectory toolbar
Toggle the display of the **Trajectory** toolbar.

Timeline
For information about the **Timeline** and the menu see chapter “Timeline control bar” on page 59.

Data info 1 2 3
Toggle the display of the **Data info** windows (maximum 3 windows) for showing and plotting 2D, 3D, 6DOF, analog and force data in both preview and file mode. For detailed information see chapter “Data info window” on page 85.

Project View
Toggle the display of the **Project view** window on the left side of the main window see chapter “Project view” on page 19.

Messages
Toggle Messages window. For detailed information see chapter “Messages window” on page 94.

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

Trajectory Editor
Open the Trajectory Editor window.

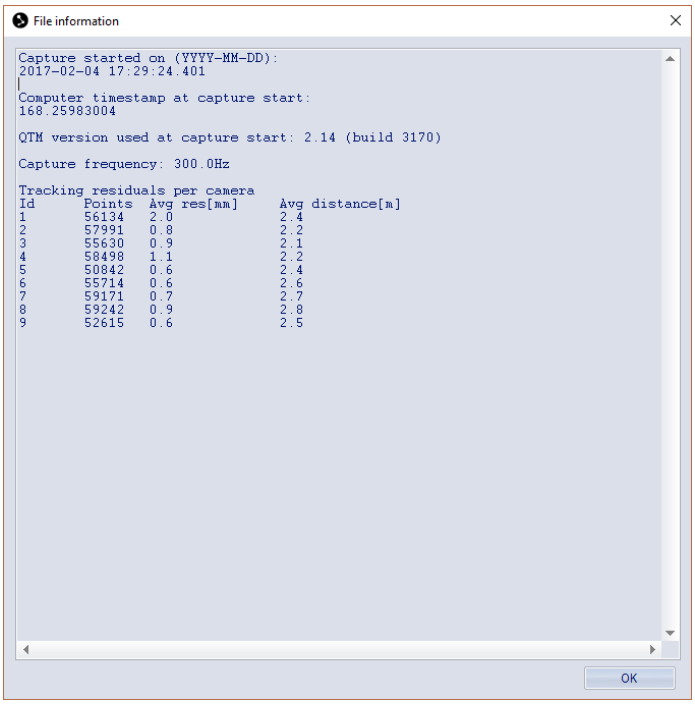
Full Screen
Enable/disable full screen mode.

File Information
Opens a window showing information about the capture, see chapter “File Information” on the facing page.

Real-Time Client Information
Opens a window showing information about currently connected real-time streaming clients and a log of real-time command and response communication.

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 57 respectively “2D view window menu” on page 39.

File Information



The File Information window shows the following information about the capture.

Start of capture as date and time according to the computer clock.

Computer timestamp at start of capture in seconds and ticks from when the computer was started.

Note: The time stamp is recalculated when trimming the file.

WARNING: The time stamps indicate the start of the capture according to the QTM software. The time stamps may not exactly correspond to the first frame of the capture and it is discouraged to use them for synchronization with data from other devices connected to the same or another synchronized computer.

QTM version used for the capture.

Capture frequency used for marker tracking in Hz.

Analog devices used for the capture and their respective capture frequencies.

Tracking information per camera, including number of points, tracking residuals and average distance to the tracked markers. For more information about the tracking residuals per camera, see chapter “3D tracking test” on page 335.

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 60.

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

Next event / Previous event
Go to next/previous event in the file. Only available if there are events in the file.

Play with Real-Time Output
The file is played with RT output until you press Stop. This feature can be used to test the RT protocol and RT plugins as it gives full control of what is sent. The speed of the real time process can be controlled with the **Playback speed** bar.

Capture

Capture
Start the motion capture, see chapter “Capturing of measurements” on page 281. 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 270.

Refine calibration
Open the **Refine calibration** dialog, see chapter “Refine calibration” on page 277.

Calibrate Twin System
Start a Twin system calibration, see chapter “Performing a Twin calibration” on page 307.

Calibrate Dikablis Eye Tracker
Start a Dikablis Eye Tracker calibration, see chapter “Dikablis gaze vector calibration” on page E - 6.

Reprocess
Reprocess the active capture file, see chapter “Reprocessing a file” on page 327.

Recalculate forces
Recalculate the force data in the active capture file, see chapter “Calculating force data” on page 386.

Linearize camera
Start linearization of a camera, see chapter “Linearize a camera” on page 263.

AIM

For information on automatic identification see chapter “Automatic Identification of Markers (AIM)” on page 364.

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.

Skeleton

For more information on skeleton calibration and tracking, see chapter “Tracking of skeletons” on page 350.

Calibrate skeletons
Create skeletons from T-pose in current frame and solve. The skeleton definition will be added to or updated in the **Skeleton solver** page in the **Project Options** and applied to the current file or real time preview.

Display Marker Set Guide
Open PDF with information about the **Qualisys Animation Marker Set**, including the markers and their placement, T-pose requirements and the use of extra markers.

Tools

Project options
Open a dialog with settings for the QTM software, see chapter “Project options dialog” on page 119.

Analyze Trajectory
Open dialog to analyze trajectories, see chapter “Analyze” on page 74.

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 98.

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 99. 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.

Open documentation folder
Open the documentation folder that contains for example the PDF version of the QTM manual and a Getting started manual.

Start Qualisys Remote Support
Start the Qualisys Remote Support program for remote support over Internet, please contact Qualisys AB on support@qualisys.se first to book a time. When the program is started support can connect to your computer and you can also chat with support.

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

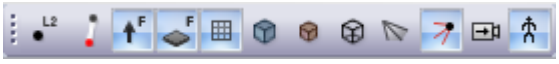
Reprocess the file

Recalculate the force data

Linearize camera

Add event

GUI Control toolbar



The **GUI Control** contains icons for toggling GUI elements in the 3D view. To change the appearances of the GUI elements open the **3D view settings** page in the **Project options** dialog. The toolbar contains the following icons, from left to right:

Marker Labels

Bones

Force Arrows

Force Plates

Grid

Covered Volume

Calibrated Volume

Bounding Box

Camera View Cones

Camera Rays

Follow Selected Markers

Skeletons

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

Calibrate skeletons

Analyze trajectory

Trajectory toolbar

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

Show/hide data info window 1

Show/hide data info window 2

Show/hide data info window 3

Show/hide project view

Show/hide messages window

Show/hide status bar

Show/hide labeled trajectories window

Show/hide unlabeled trajectories window

Show/hide discarded trajectories window

Show/hide trajectory editor window

Keyboard shortcuts

Menu shortcuts

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

Workflow shortcuts

Ctrl + N
New capture file.

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

Ctrl + Shift + Alt + C
Open the **Start calibration** dialog.

Ctrl + Shift + R
Open the **File reprocessing** dialog.

F9
Apply AIM model to a capture file or restart AIM and 6DOF calculation in preview and capture mode.

F10
Calibrate skeletons.

Capture file shortcuts

Ctrl + O
Open capture file.

Ctrl + S
Save capture file.

Ctrl + Shift + S or **F12**
Save as a new capture file.

Ctrl + F4
Close capture file.

Ctrl + F
Find a file in the project.

F3
Find next file in the project.

Editing shortcuts

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.

Del
Delete selected bones or "degrade" selected trajectories:

Labeled trajectories: move the selected trajectories/parts to **Unidentified trajectories** window.

Unidentified trajectories: move the selected trajectories/parts to **Discarded trajectories** window.

Discarded trajectories: permanently delete the selected trajectories/parts.

Ctrl + Shift + T
Trim a capture file to the current measurement range.

Display and window shortcuts

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

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

Ctrl + R
Toggle the display of the **Project view** window.

Ctrl + T
Toggle the display of the **Trajectory Editor** 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.

F11 or **Shift + Alt + Enter**
Enable/disable full screen mode.

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.

Shift + M
Activate the **Marker mask** tool.

P
Toggle 3D view projection between orthogonal and perspective.

Ctrl + Alt + Arrow
Switch between orthogonal viewpoints

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 selected bones or "degrade" selected trajectories:

Labeled trajectories: move the selected trajectories/parts to **Unidentified trajectories** window.

Unidentified trajectories: move the selected trajectories/parts to **Discarded trajectories** window.

X
Split the selected trajectories after the current frame.

Ctrl + Alt
Quick identification method, see chapter “Manual identification of trajectories” on page 362.

Shift + B
Activate the **Create bones sequence** tool.

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.

Ctrl + drag
Scrubbing feature (3D view window). Hold control key, press left mouse button anywhere in the empty 3D space and drag to scrub through the measurement's time line.

Shift + Mouse wheel
Trace range zooming feature. In the 3D view window, hold the shift key and use the mouse wheel to increase or decrease the trace range.

F8, Shift + F8
Use selected trajectories to create a new rigid body definition, based on average of frames (**F8**) or on the current frame (**Shift + F8**).

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.

J and **Shift + J**
Jump in time to the next unidentified trajectory or part, respectively, and center on it in the **3D view** window.

S
Swap the selected parts of two trajectories.

W
Swap the part of the current frame in two selected trajectories.

X
Split the selected trajectories after the current frame.

Shift + F8
Define a rigid body with the current frame of the selected trajectories.

F8
Define a rigid body with an average of data in all frames of the selected trajectories.

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

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

D
Move the selected trajectories/parts to the **Discarded trajectories** window.

Del
Delete selected bones or "degrade" selected trajectories:

Labeled trajectories: move the selected trajectories/parts to **Unidentified trajectories** window.

Unidentified trajectories: move the selected trajectories/parts to **Discarded trajectories** window.

Discarded trajectories: permanently delete the selected trajectories/parts.

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.

Ctrl + Shift + Space
Start playing with real time output

Ctrl + drag
Scrubbing feature (3D view window). Hold control key, press left mouse button anywhere in the empty 3D space and drag to scrub through the measurement's time line.

Ctrl + G
Go to frame.

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.

Page Down
Go to next event.

Page Up
Go to previous event.

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**.

Trajectory Editor shortcuts

The following keyboard shortcuts and mouse gestures can be used in the **Trajectory Editor** window.

Keyboard shortcuts:

Ctrl + A (select all)
Select complete horizontal range.

Delete (delete)
Delete selected range.

F (fill gap(s))
Fill selected gap(s) using current fill type.

C (center current frame)
Center visual range on current frame.

Z (zoom to selection)
Zoom to the currently selected range.

X (zoom to extents)
Zoom horizontal and vertical axis to data extents.

H (zoom horizontal)
Zoom horizontal axis to data extents.

V (zoom vertical)
Zoom vertical axis to data extents.

A (auto zoom vertical)
Toggle zoom vertical axis automatically.

B (auto zoom horizontal)
Toggle zoom horizontal axis automatically when selecting trajectories.

G (zoom to gap)
Toggle automatic zoom to selected gap.

T (time mode)
Toggle time units in seconds/frames.

Q, **Shift + Q**
Traverse the trajectory list down and up, respectively.

Mouse gestures:

Left mouse button - double click
Set current frame.

Left mouse button - drag
Select range.

Right mouse button - click
Open **Trajectory Editor** window menu.

Right mouse button - drag
Pan.

Middle mouse button - click
Toggle split/merge view.

Ctrl + Left mouse button - drag
Scrub timeline / Move current frame.

Alt + Left mouse button - drag
Box zoom.

Mouse wheel
Horizontal zoom.

Shift + Mouse wheel
Horizontal pan.

Ctrl + Mouse wheel
Vertical pan.

Note: There may be overlapping shortcuts for the 3D/2D view. Make sure that the **Trajectory Editor** window is in focus for the above to apply.

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 270.

Calibration results dialog
This dialog shows the results of a calibration, see chapter “Calibration results” on page 271.

Recalibration settings dialog
This dialog is used to recalibrate a capture file, see chapter “Reprocess calibration” on page 279.

Start capture dialog
This dialog is used to start a capture, see chapter “Start capture” on page 282.

File reprocessing dialog
This dialog is used to reprocess a capture file, see chapter “Reprocessing a file” on page 327.

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: “Input Devices” on page 121, “Processing” on page 181, “GUI” on page 241 and “Miscellaneous” on page 250. 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.

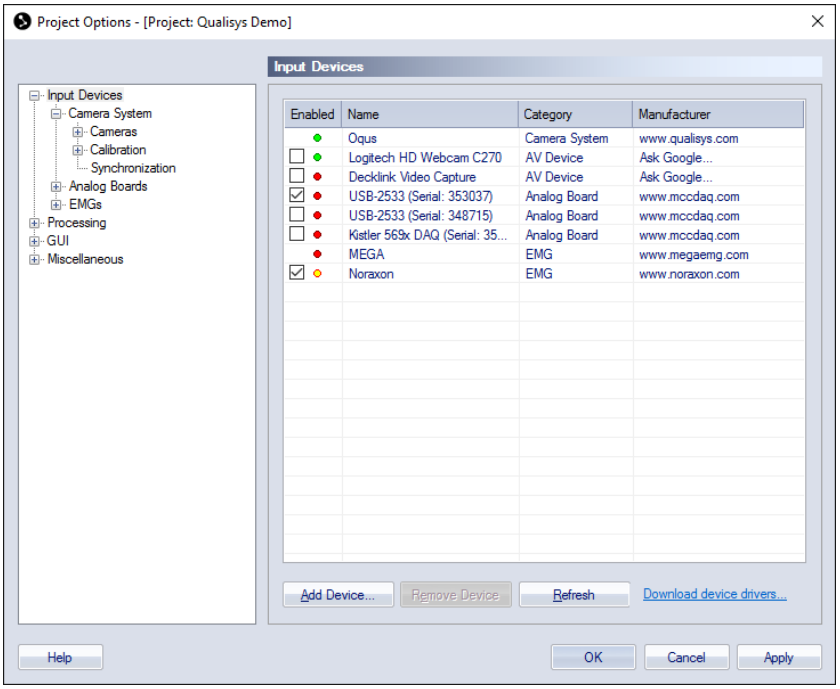
Click **Cancel** to close the dialog without saving the options.

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

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 327.

Project options

Input Devices



The **Input Devices** page lists the available devices in QTM. Double-click on an entry to go to the settings for that device. The list contains the following information:

Enabled
The **Enabled** column contains the checkbox to enable the device and also a status light for the device. A green circle means that the device is connected, a yellow circle means that the device status is unknown and red circle means that the device is disconnected.

Note: Devices where the connection status is unknown are always displayed in the list.

Name
The **Name** of the device.

Category
The **Category** of the device. The different categories are:

Camera system
Qualisys systems.

Analog board
Analog boards supported by QTM.

Note: The analog board settings are saved with the serial number of the board. So the analog board will be visible in the project even if you disconnect the board from the computer or it is not active in Instacal.

AV devices
Any video camera connected to the computer, for example via a Blackmagic Design card or a webcam.

Force plate
Digitally integrated force plates.

EMG
Digitally integrated EMGs, see chapter “EMG systems in QTM” on page D - 1.

Eye Tracker
Digitally integrated eye trackers, see chapter “Eye tracking hardware in QTM” on page E - 1.

Manufacturer
The **Manufacturer** column contains a link to the website of the device's manufacturer. If the manufacturer is unknown there is a link to Google.

There are four options at the bottom of the list:

Add Device
There are three devices that needs to added to the devices list: Delsys, Dikablis and SMI. See the respective chapter for more information: “Trigno installation” on page D - 26, “Add Dikablis eye tracker to a project” on page E - 2 and “Add SMI eye tracker to a project” on page E - 16.

Remove Device
You can remove any disconnected (red circle) device.

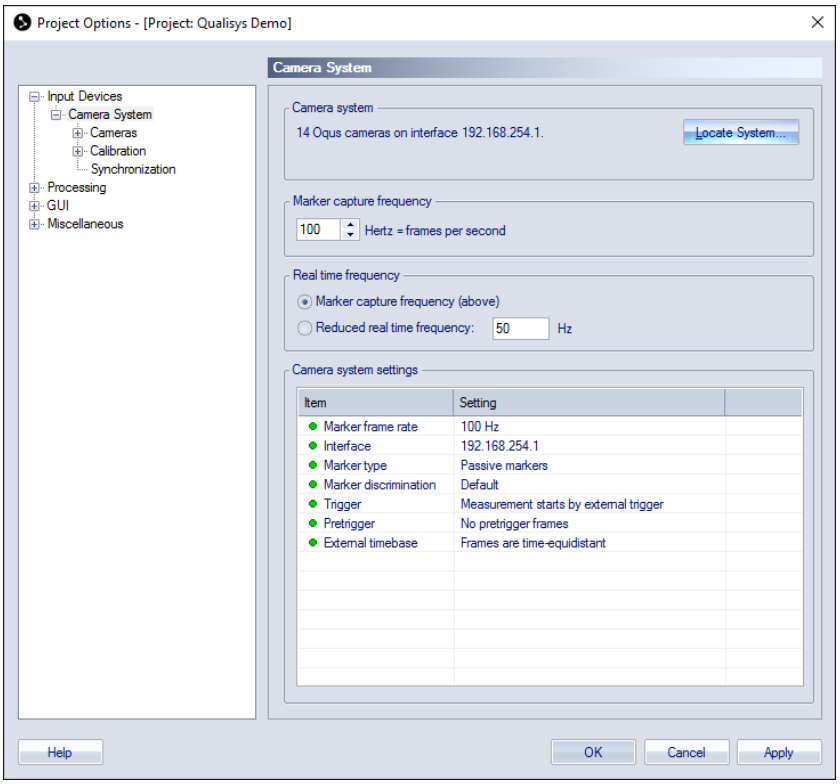
Refresh
Use the **Refresh** button to refresh the status of the devices.
Note: The refresh button does not work on analog boards when they are added or changed in Instacal. In that case you need to restart QTM.

Download device drivers
The **Download device drivers** link opens a website with current drivers for Delsys, AMTI and SMI.

Camera system

The **Camera system** branch of the options tree consists of hardware settings for the cameras. In the **Camera system** page you can locate the camera system (**Locate System**), set the **Marker capture frequency** and the **Real time frequency**, and obtain an overview list with the status of the **Camera system settings**.

The **Camera system** page contains the following subcategories: **Cameras**, **Calibration** and **Synchronization**.



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 an overview of maximum capture frequencies per camera, see “Qualisys camera sensor specifications (marker mode)” on page A - 1.

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 313. 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 an 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 159.

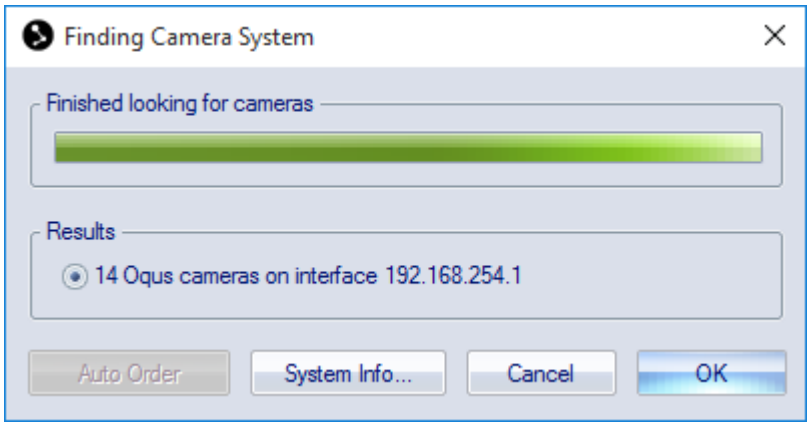
Click on an entry to go directly to the corresponding page in the **Project options** dialog. By right-clicking on an entry a menu with two options appear: **Change** and **Reset to default(s)**. Click **Change** to go directly to the corresponding page. Click **Reset to default(s)** to reset that setting to the default value.

Locate system

Under the **Camera system** heading the currently located system is listed. To locate the system use the **Locate system** button. QTM will open the **Finding camera system** dialog and start looking for the system.

Finding camera system

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

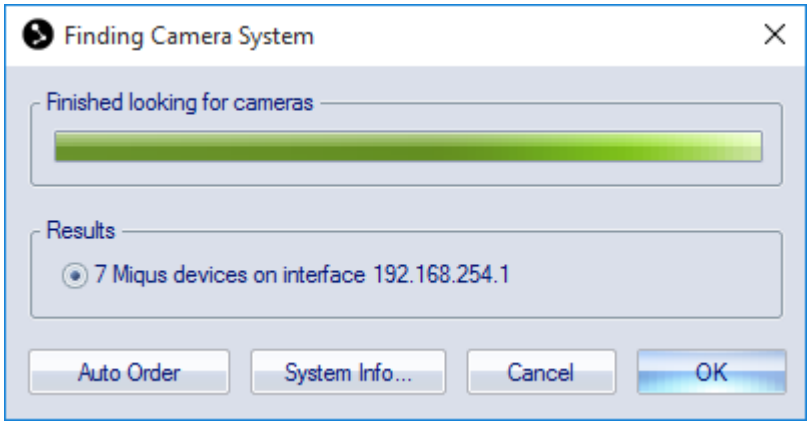


Note: If one or more devices have old firmware, a dialog appears for applying a necessary firmware update for using the system with the installed version of QTM. For more information see chapter “Firmware update when locating system” on page 443.

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

Click on **System info** to see detailed information about the selected system. For Miqus systems the **Auto order** button can be used to automatically order the cameras in the 2D View window.

Auto order (Miqus)

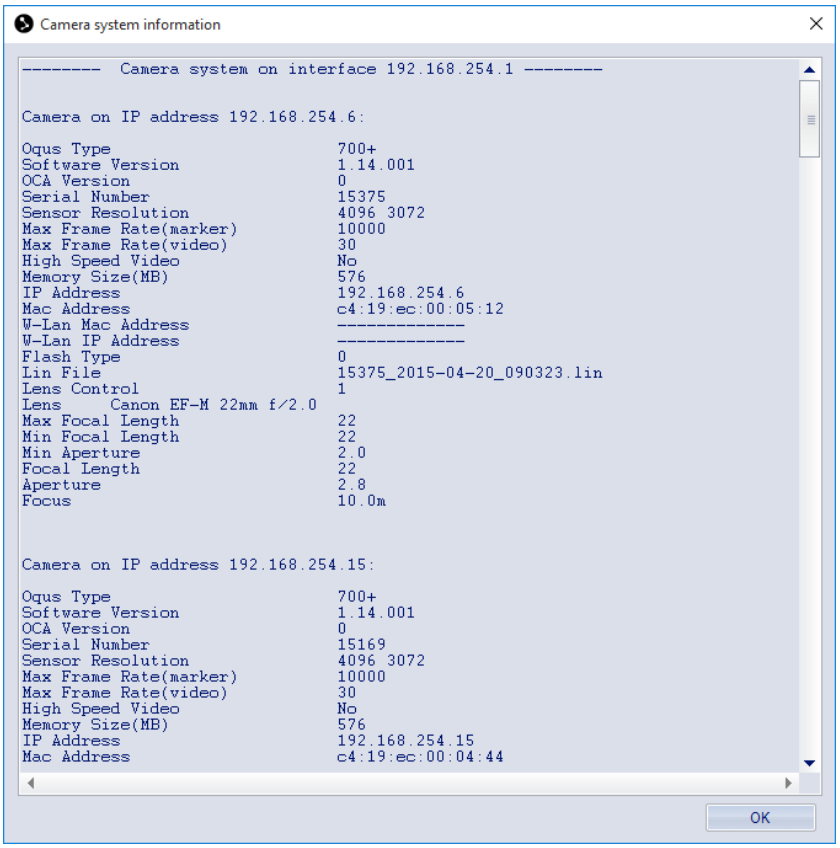


When a Miqus system is selected, it is possible to automatically order the cameras by pressing the **Auto Order** button. When the ordering is finished, the **Auto Order** button is renamed to **Reverse Order**. For more information about the ordering of Miqus cameras, see “Automatic ordering of the cameras (Miqus)” on page 256.

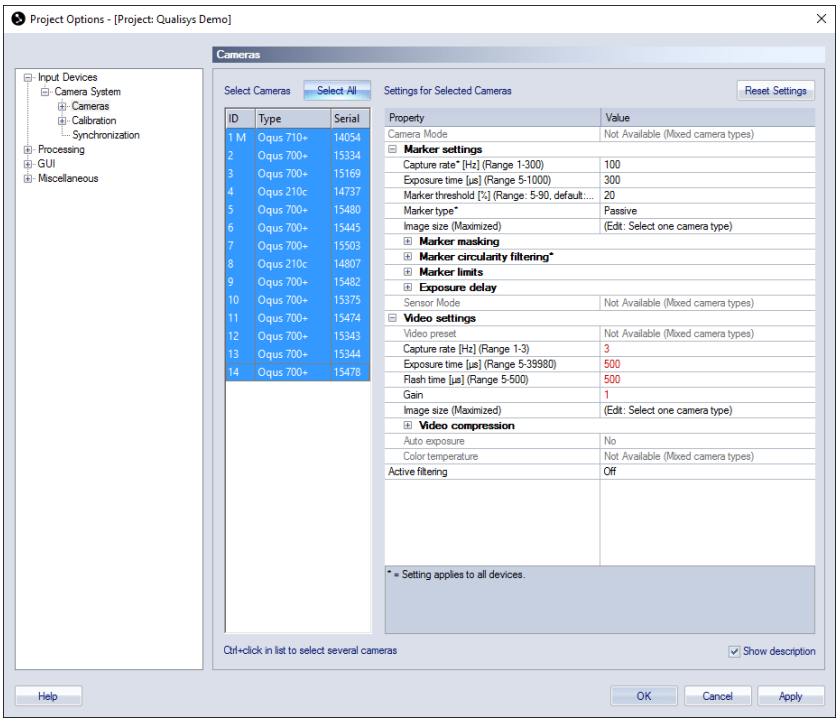
Camera system information

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

The **Camera system information** dialog shows information about the individual Qualisys devices as illustrated below. This is the same information as **Camera information** in the **2D view window** menu.


Camera system information
-------- Camera system on interface 192.168.254.1 --------
Camera on IP address 192.168.254.6:
Oqus Type 700+
Software Version 1.14.001
OCA Version 0
Serial Number 15375
Sensor Resolution 4096 3072
Max Frame Rate(marker) 10000
Max Frame Rate(video) 30
High Speed Video No
Memory Size(MB) 576
IP Address 192.168.254.6
Mac Address c4:19:ec:00:05:12
W-Lan Mac Address ----------
W-Lan IP Address ----------
Flash Type 0
Lin File 15375_2015-04-20_090323.lin
Lens Control 1
Lens Canon EF-M 22mm f/2.0
Max Focal Length 22
Min Focal Length 22
Min Aperture 2.0
Focal Length 22
Aperture 2.8
Focus 10.0m
Camera on IP address 192.168.254.15:
Oqus Type 700+
Software Version 1.14.001
OCA Version 0
Serial Number 15169
Sensor Resolution 4096 3072
Max Frame Rate(marker) 10000
Max Frame Rate(video) 30
High Speed Video No
Memory Size(MB) 576
IP Address 192.168.254.15
Mac Address c4:19:ec:00:04:44
OK

Cameras



The **Cameras** page contains all of the camera settings for the Qualisys cameras. It is often easier to use the tools in the 2D view and the **Camera settings** sidebar if you want to change settings that are used more often, see chapters “2D view toolbar” on page 29 and “Camera settings sidebar” on page 31.

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, see chapters “Camera Mode” on the next page, “Marker settings” on the next page, “Video settings” on page 134 and “Active filtering (Oqus)” on page 139.

The camera list has three 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.

Serial
The serial number of the camera.

On the **Cameras** 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 values, 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 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.

Camera settings

Camera Mode

Camera Mode	Marker

The **Camera Mode** setting switches the camera mode for the selected cameras. The available modes are **Marker**, **Marker Intensity** and **Video**. The modes can also be changed during preview in the 2D view. Changing the modes in the 2D view will update the **Camera Mode** setting.

When starting a measurement, the cameras will start in the mode selected with the **Camera Mode** setting. However, since changing in the 2D view updates the setting it means that it is usually the same as the last preview.

Marker settings

The **Marker settings** include all of the settings for marker mode. The settings are described below.

Capture rate

Capture rate* [Hz] (Range 1-187)	100

The **Capture rate** is the frequency that is used during a marker measurement. The setting is global for all cameras in Marker mode.

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 camera types within the system, as well as the size that is set with **Image size**. For an overview of the maximum capture frequencies per camera model at full size image, see “Qualisys camera sensor specifications (marker mode)” on page A - 1.

Exposure time

Exposure time [µs] (Range 5-1000)	300

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) or at most 1000 µs in marker mode.

Note: For Qualisys underwater cameras exposure times longer than one tenth of the period can be used. Avoid the use of long exposure times when testing the cameras in air because they may overheat.

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 258.

Marker threshold

Marker threshold [%] (Range: 5-90, default:...	17

The **Marker threshold** sets the threshold intensity level for markers in Qualisys cameras. The setting can be set individually for each camera.

The level can be set between 5 and 90 % the max intensity level, the default value is 17. Every pixel value that is brighter than the specified value will be calculated as a 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 258.

Marker type

Marker type*	Passive markers

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
Use this setting for any marker with reflective material.

Active markers
Use this setting for the short range active marker with sequential coding.

Passive + 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. However, if you mix the passive and active markers all the time you lose 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.

Untriggered active markers
Use this mode for active markers that are lit constantly. In this mode the strobe is turned off to minimize unwanted reflections.

Inverted passive markers
This mode is only available for the 3+-series. It inverts the marker detection so dark areas are calculates as markers instead of the bright areas.

For information about passive and active markers, see chapter “Passive vs Active markers” on page 261. For information about the short range active marker, see chapter “Short range active marker” on page 445.

Note: The old type of close range active marker is not supported in QTM 2.5 or later. Please contact Qualisys AB if you use this type of active marker.

Image size

Image size (Selected are maximized)	0, 1279, 0, 1023
Left (Default: 0, Steps of 32)	0
Right (Max: 1279, Steps of 16)	1279
Top	0
Bottom (Max: 1023)	1023

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 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 camera type. The steps for the current model is displayed next to the settings. The maximum size depends on the camera type and the selected sensor mode.

By reducing the image size it is possible to increase the capture frequency. The frequency range will be updated automatically when you change the size. The image size is showed with a 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. In the 2D view of a file the aspect ratio of the camera view is adapted to the reduced image size.

Note: The red rectangle representing the image size is not drawn linearized, this means that with wide angle lens it is best to turn off the **Show linearized data** option on the **2D view settings** page to see the true positions of the mask and image size.

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 2c-, 5+- and 7+-series cameras is only limited in the y-direction when increasing the capture rate. However when increasing the capture rate on the **Camera settings** sidebar then the image ratio is kept. You can then increase the width of the image size by dragging on the red rectangle.

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 to the camera image to discard markers in that area of the image. 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 297.

Note: Software marker masks can be applied to a file, see chapter “How to use software marker masks” on page 331.

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 298.

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 Oqus and 20 masks per Miqus. 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.

Note: Marker masks are not drawn linearized, this means that with wide angle lens it is best to turn off the **Show linearized data** option on the **2D view settings** page to see the true positions of the mask and image size.

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 (Oqus)

Marker circularity filtering	
Enable*	☑
Circularity threshold	Medium (default)

For Oqus cameras the option to use marker circularity filtering is available. The **Marker circularity filtering** options are used to filter non-circular markers in the 2D data. The default is that the filtering is not **Enabled**. For more information about the filtering see chapter "How to use non-circular marker settings" on page 299.

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 has not been able to process are considered to be OK.

The marker filtering is done in the camera according to the options below.

Enabled
To turn on the filtering check the **Enabled** checkbox. Which markers that are filtered out depends on the **Circularity level** option.

Circularity level
The **Circularity level** option defines which markers are filtered out as too non-circular. These markers are then processed according to the **Non-circularity marker settings** on the **2D preprocessing and filtering** page.
The option has five levels: All markers, Low, Medium, High and Very high. The default value is **Medium**. When set to **All markers**, then all markers are considered to be non-circular and all segments are therefore, if possible, sent to QTM. The levels Low to Very high corresponds to a circularity relation of 20 to 80% between x- and y-size.

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 128 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.

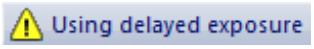
Exposure delay

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 it may be harder to get a good tracking. For more information see chapter "Delayed exposure to reduce reflections from other cameras" on page 259.

Note: Delayed exposure does not work on 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	1.3 MP @ 502 Hz

The **Sensor mode** option can be used to reduce the resolution to get a higher capture ratewhile keeping the same field of view. The option is only available for camera models that have more than one sensor mode. The setting can be changed individually on each camera and also for Marker and Video mode separately. The sensor mode setting can also be changed on the **Camera settings** sidebar. For an overview of available sensor modes per camera model, see “Qualisys camera sensor specifications (marker mode)” on page A - 1.

Video settings

The **Video settings** contains all of the settings for video mode. The settings are described below.

Capture rate

Capture rate [Hz] (Range 1-187)	100

The **Capture rate** on the **Video settings** is only used for video capture and can be set independently of the marker capture rate. The setting is also individual for the cameras so that you can capture at different video capture frequencies.

The maximum capture rate can be increased by reducing the **Image size**. 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 when choosing a higher frequency.

For an overview of maximum capture rates for each camera model, see “Oqus high-speed video sensor and memory specifications” on page A - 2 for uncompressed video and “Qualisys video sensor specifications (in-camera MJPEG)” on page A - 3 when using in-camera MJPEG compression.

Note: It is possible to capture short video sequences with Qualisys cameras that are not configured for high-speed video. The capture rate is then limited to 30 Hz, even if you use a reduced image size.

Exposure time

Exposure time [microsecs] (Range 5-3333)	2000

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 microseconds less than the period time, i.e. 1/capture rate.

Flash time

Flash time [microsecs] (Range 5-2000)	5

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 µs.
The IR flash does not contribute much when you capture video. Therefore, it is often a good idea to set the **Flash time** to its minimum of 5 µs.

Gain

Gain	1

With the **Gain** option the intensity of the video image can be increased. Select the desired gain from the drop-down list. The available values are 1, 2, 4, 8 and 16 (for older camera models the maximum gain is 4). The default value is 1 except for the Oqus 2c-series and Miqus Video where it is 4, or it is set automatically when using **auto-exposure**. Use the **Gain** option if the exposure time and aperture is already at their maximums.

All cameras, except the Oqus 3 and 5 series, have analog gain before the image capture and therefore the image quality is not decreased at the same degree with increasing gain. The number of gray scales in the image is also the same for different gain settings.

The Oqus 3 and 5-series have 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 gray scales are decreased.

Note: The **Gain** option is available on the **Camera settings** sidebar for all cameras except the Oqus 3- and 5-series.

Image size

Image size (Selected are maximized)	0, 1279, 0, 1023
Left (Default: 0, Steps of 32)	0
Right (Max: 1279, Steps of 16)	1279
Top	0
Bottom (Max: 1023)	1023

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 camera type. The step size for the current model is displayed next to the settings. For an overview of the maximum image size for each camera model, see “Oqus high-speed video sensor and memory specifications” on page A - 2 and “Qualisys video sensor specifications (in-camera MJPEG)” on page A - 3.

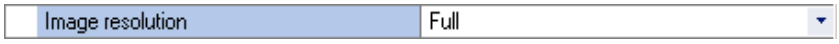
Note: The x size of the Oqus 3 series camera must be at least 160 pixels. The x size of the Oqus 3+ series camera must be at least 144 pixels in **Standard** mode and 72 pixels in **High-speed** mode.

Note: In the uncompressed mode of Oqus 2c, 5+, 6+ and 7+ series cameras the image size is only limited in the y-direction when increasing the capture rate. However, when increasing the capture rate on the **Camera settings** sidebar, the image ratio is kept constant. You can then increase the width of the image size by dragging on the red rectangle.

Note: The image size of the Oqus 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.

Note: The **In-camera MJPEG** mode for the Oqus 5+-series has a larger maximum image size because it compresses the image in the camera.

Image resolution



The image resolution will change the quality of the images that are captured by Qualisys cameras according to the options below. The setting can be set individually for each camera.

Note: The setting is not available for the Oqus 2c, 5+ and 7+-series cameras. If you want to reduce the image size on those cameras you have to use the Sensor mode setting, which has the advantage of also increasing the maximum capture rate. For more information, see chapter “Sensor mode” on page 139.

Note: The Oqus 4-series camera cannot handle full resolution for video images, therefore the maximum image resolution is 1/4.

Note: To optimize the memory usage there is a column of black pixels to the right in the image. This 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	
Mode	In-camera MJPEG
Compression quality [%]	80

The **Video compression** option can be used to reduce the video file size. By default the **Mode** is set to **In-camera MJPEG** for all cameras that support that option (i.e. Oqus 2c, 5+ and 7+ series, as well as Miqus Video). You can set the **Compression quality** of the MJPEG compression to change the quality and file size of the video; the default is 50. Increasing the quality will lead to larger images and in some cases this means that not all of the video images can be sent during the capture. These frames will be fetched after the capture is finished.

Note: For Miqus Video, the Video compression setting is not available. Miqus Video cameras always stream video using In-camera MJPEG.

Video compression	
Mode	None

The default mode on the other cameras is **None** for uncompressed image and maximum quality. 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.

Switch the **Mode** setting to **Software** to specify a codec on the computer for compression of the video.

Video compression	
Mode	Software
Compression codec	ffdshow Video Codec
Configure codec	Press the button for configuration
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 codecs are then grouped in **Recommended codecs** and **Other codecs**. The recommended codecs are **ffdshow Video Codec** 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 291.

Note: If none of these codecs are installed on the computer there are no alternatives under the **Recommended codecs** heading. You can install the codecs by re-installing QTM and selecting the **FFDshow MJPEG Codec** option during installation.

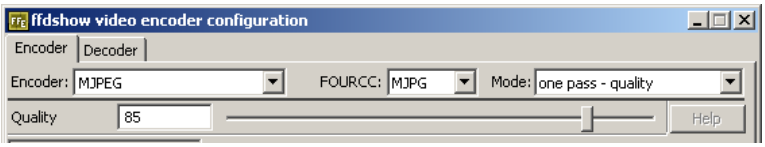
The settings for a codec are opened with the ··· button on the **Configure codec** line. The settings depend on the codec, please refer to the codec instructions for help on the settings. The FFDS codec is recommended with the MJPEG codec and it is important to set the settings according to the chapter “FFDS-MJPEG codec settings” on the 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.

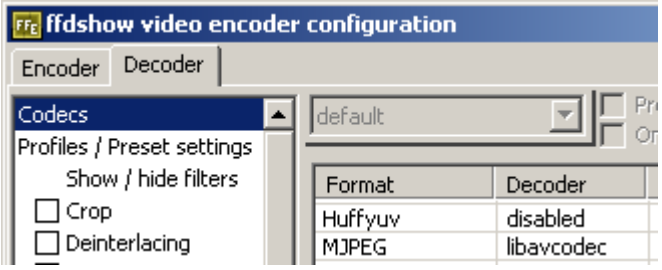
Click on the ··· button on the **Codecs recommended by Qualisys** line to go to a web site with links to codecs.

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). The file size can be reduced by choosing a lower quality (more compression).



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/Grayscale

Color/Grayscale	Color

The **Color/Grayscale** option is only available if you have connected an Oqus 5-series 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.

Grayscale
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.

Color
Use bilinear color interpolation to get a color image.

Note: If you want to use the color interpolation on 5-series with the **Image resolution** option, then you must select 1/9 or 1/36 for the color interpolation to work. The other options delete the color information from the sensor.

Auto exposure

Auto exposure	Yes
Auto exposure compensation [EV]	0
⊞ Auto exposure area (Maximized)	0, 0, 1919, 1087

The **Auto exposure** option is available on Miqus Video and Oqus 2c cameras. With the auto exposure the exposure time and gain is controlled automatically so that image is

bright enough. When the camera is using one of the predefined presets then the auto exposure is always on. For the **Custom** preset it must be turned on with the **Auto exposure** option.

When turned on there are two additional settings **Auto exposure compensation [EV]** and **Auto exposure area**. The Auto exposure compensation option controls how bright the image will be. Increase the value to have a brighter image and decrease it to get a darker image. This option is also available in the **Camera settings** sidebar in preview.

By default the whole image area is used to calculate the brightness of the image. The area used in the brightness calculation can be set with the **Auto exposure area** option. In this way the camera can calculate the brightness on the part of the image that is most useful. This can be used for example when there is a bright window in the image. It is easiest to define the area with the **Auto Exposure** tool in preview, see chapter "2D view toolbar" on page 29.

Color temperature

Color temperature	Daylight

The **Color temperature** setting is only available for the Oqus Color video (2c) camera. With it you can adapt the color temperature used for the white balance in the video. If you use the wrong option the colors may be very wrong, so it is important to test the different options to find the best one.

Daylight
This option uses a color temperature that is most suitable for daylight. It works best outdoors, but also works well if you have large windows.

Office lighting (Cool white fluorescent)
This option is for bluer type of lights. It is usually not the best option if you mixed types of lighting, for example if there are windows.

Home lighting (Incandescent)
This option is for more yellow type of lights, like light bulbs. It is usually not the best option if you mixed types of lighting, for example if there are windows.

No color correction
For this option the color temperature is not adjusted at all. It can for example work well in situations with mixed types of light.

Sensor mode

Sensor Mode	1.3 MP @ 502 Hz

The **Sensor mode** option can be used to reduce the resolution to get a higher capture rate while keeping the same field of view. The option is only available for camera models that have more than one sensor mode. The setting can be changed individually on each camera and also for Marker and Video mode separately. The sensor mode setting can also be changed on the **Camera settings** sidebar. For an overview of available sensor modes per camera model, see "Oqus high-speed video sensor and memory specifications" on page A - 2 and "Qualisys video sensor specifications (in-camera MJPEG)" on page A - 3.

Active filtering (Oqus)

Active filtering	Off

The **Active filtering** mode is used to improve capturing in daylight conditions. It is available on the Oqus 1, 3+, 4, 5+, and 7+ 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 296.

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 Oqus 3+ series the maximum frame rate is 244 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.

Lens aperture

Lens aperture	4.000

The **Lens aperture** setting is only available for Qualisys cameras with a motorized lens. The limits of the aperture depends on the mounted lens. In most cases it is recommended to have the aperture around 4, for more tips on aperture and focus see chapter "Setting the aperture and focus" on page 434.

It is best to change this setting in preview from the **Camera settings** sidebar in the 2D view, so that you can check the brightness and focus of the image directly.

Note: Once focus and aperture have been set for the cameras in a fixed camera system it is possible to disable lens control using the Qualisys Firmware Installer. This way the current lens settings will be fixed. For more information, see "Appendix B: Qualisys Firmware Installer (QFI)".

Lens focus distance

Lens focus distance [m]	10.000

The **Lens focus distance** setting is only available for Qualisys cameras with a motorized lens. The limits of the setting is 0.5 m and 20 m, for more tips on aperture and focus see chapter "Setting the aperture and focus" on page 434.

It is best to change this setting in preview from the **Camera settings** sidebar in the 2D view, so that you can check the focus of the image directly.

Note: Once focus and aperture have been set for the cameras in a fixed camera system it is possible to disable lens control using the Qualisys Firmware Installer. This way the current lens settings will be fixed. For more information, see "Appendix B: Qualisys Firmware Installer (QFI)".

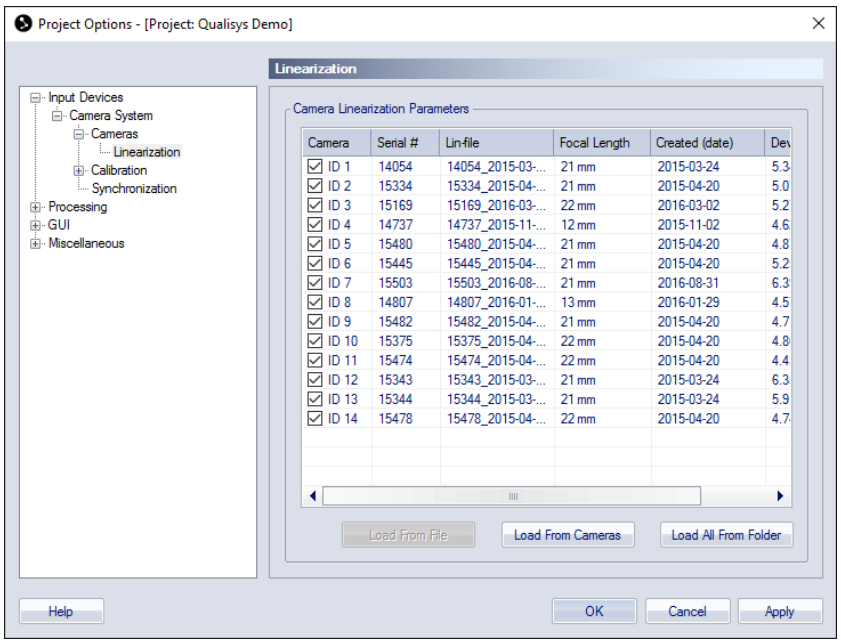
Linearization

The **Linearization** page contains information about the linearization of each camera, see chapter “Linearization of the cameras” on page 263. 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 of the connected cameras. In the list you can manage 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 327. However, if a camera has been deactivated during calibration, the calibration must be reprocessed first, see chapter “Reprocess calibration” on page 279.



The following information is displayed for each linearization file:

Camera
The ID of the camera in the current system.

Serial #
The serial number of the camera.

Lin-file
Name of the linearization file.

Focal Length
The focal length reported in the linearization file.

Created (date)
Date of creation for the linearization file.

Deviation (CU)
The deviation reported in the linearization file. This is usually a number between 2-4 subpixels (CU).

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

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

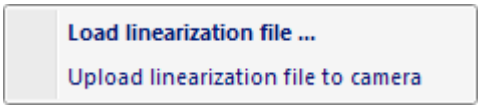
Load from cameras

Load from folder

Manually from the computer

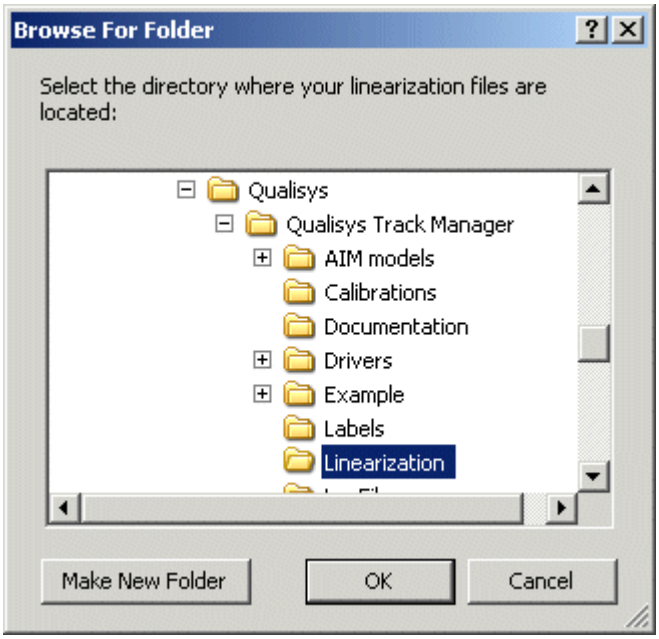
The methods are described below. The recommended method is to **Load from cameras**.

1. Click on **Load from cameras**. A dialog appears and QTM starts to load the linearization files from the Qualisys 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.

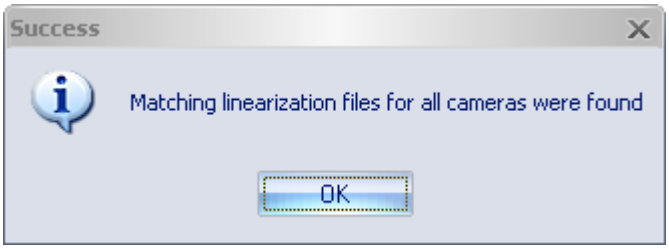
   Select **Upload linearization file to camera** to store the current file in the list in the camera. This can be used if you for example performed more than one linearization and don't want to use the last one.

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

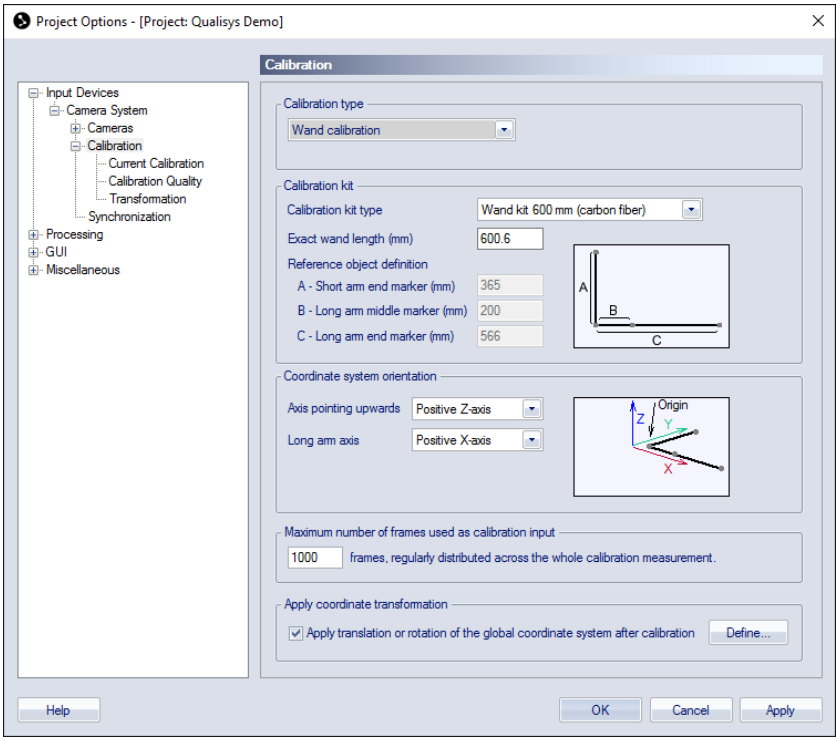
The **Calibration** page contains calibration settings required to perform a correct calibration. The lay-out and available options depend on the **Calibration type**. For all calibration types it is very important that the calibration data is entered correctly in order to obtain high quality motion capture data. For a detailed description on how to perform a calibration see chapter “Calibration of the camera system” on page 269.

Calibration type

Select the calibration type that will be used. The supported calibration types are **Wand calibration** and **Fixed camera calibration**, see descriptions 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 275.



Calibration kit

Define the calibration kit you are using under the **Calibration kit** heading. 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**:

Active wand kit 1011 mm

Wand kit 110 mm

Wand kit 120 mm

Wand kit 300 mm

Wand kit 300 mm (carbon fiber)

Wand kit 600 mm (carbon fiber), see chapter "Carbon fiber wand kit (600 mm)" on page 278.

Wand kit 750 mm

Kit defined below
The requirements for the configuration of a custom reference object (L-frame) are:

- The distance A should be shorter than the distance C.
- The distance B should be less than half the distance C.

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.

Note: The default origin of the coordinate system for the 300 mm and 600 mm carbon fiber kits, as well as the 1011 active wand kit, is at the corner where the frame rests on the floor. For the other kits the default origin is at the center of the corner marker, for information on how to translate the origin see chapter "Translating origin to the floor" on page 278.

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.

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. You can increase this value to as many frames as you like, but when you use more than about 10000 frames you will usually start noticing a longer processing time for the calibration.

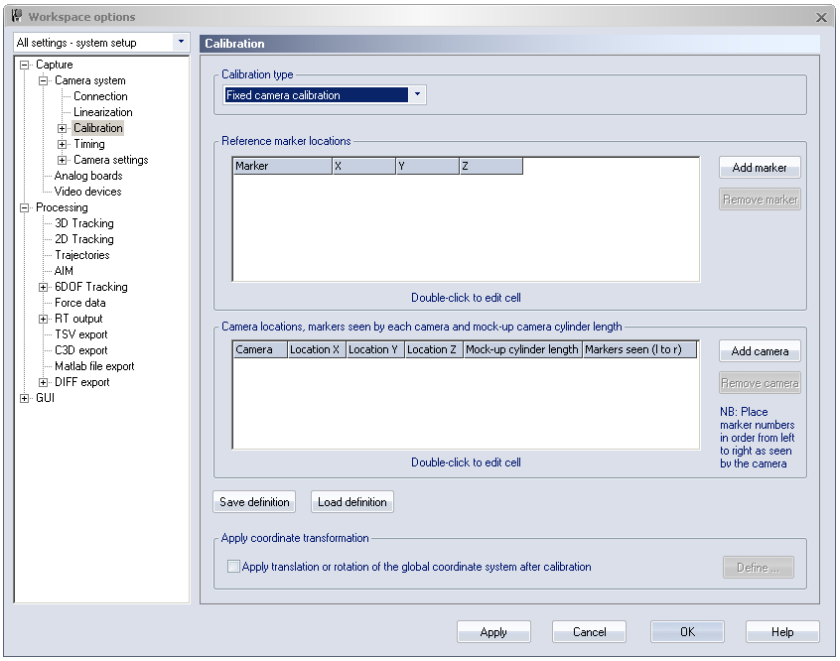
For more information about the calibration time and the frequency that are used see chapter “Calibration dialog” on page 270.

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 149.

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. 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.

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).

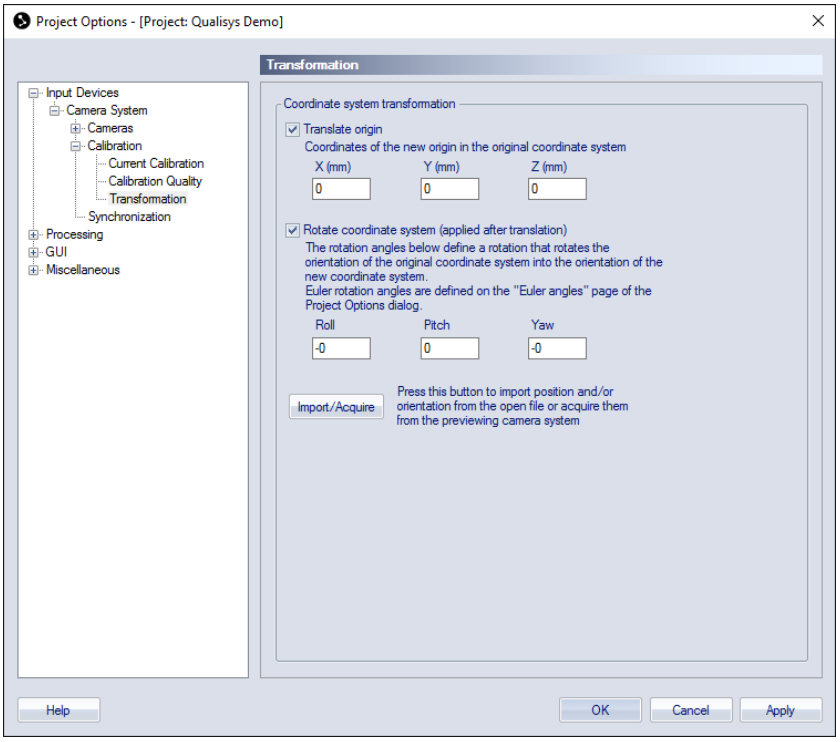
Note: QTM uses the top markers in the **2D view** window as reference markers.

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 the facing page.

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 206. The rotations are made around the following axes: Roll - X axis, Pitch - Y axis and Yaw - Z axis.

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.

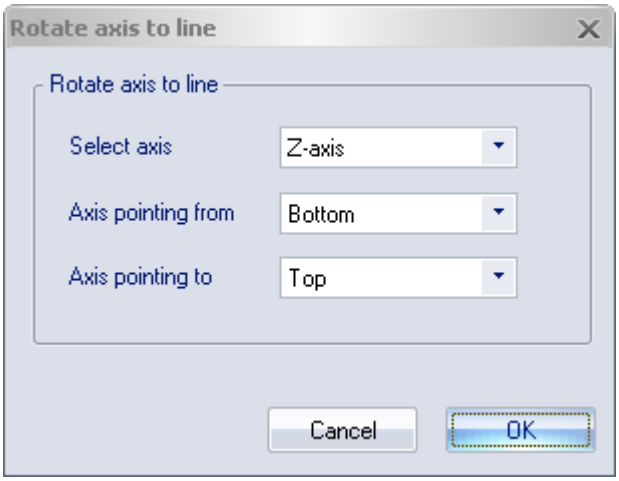
Note: The rotation angles are used to calculate a rotation matrix, which is then used to transform the coordinate system. When the resulting rotation matrix is converted back to rotation angles again it is not necessarily the same angles. This means that the angles that you have entered can change after 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.



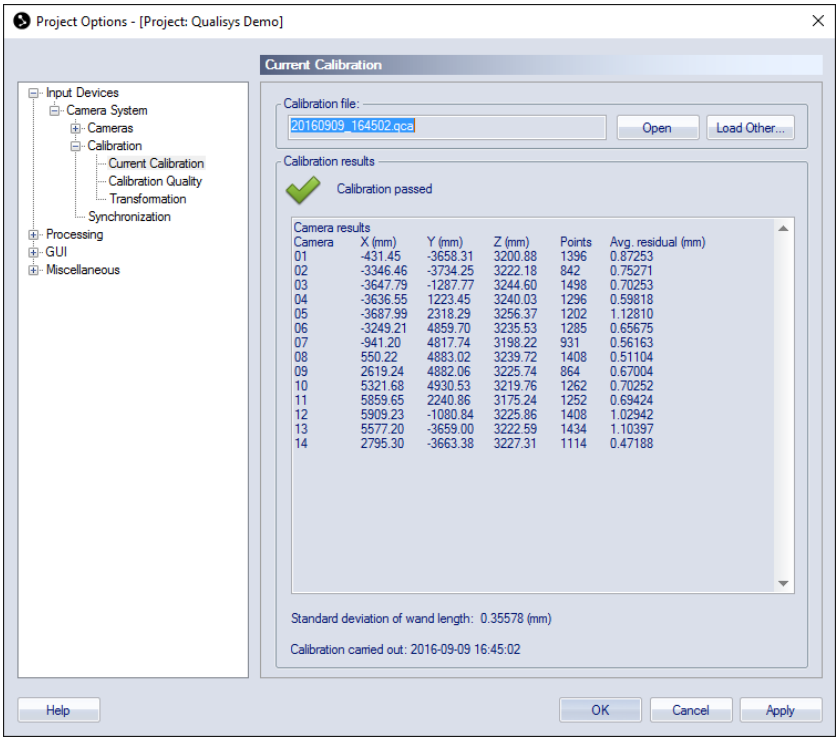
5. Select the axis that you want to define the rotation for.
6. Then select the trajectory that the **Axis is pointing from** and the trajectory that the **Axis pointing to**.

   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.

Current calibration

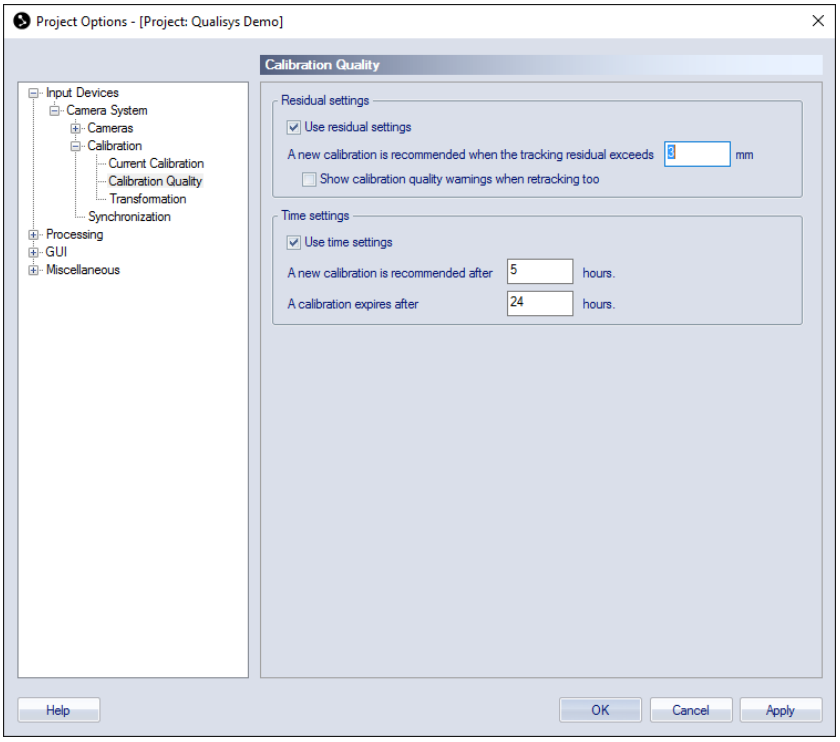


The **Current calibration** page displays the calibration that is used by QTM. Open the current calibration with the **Open** button. With the **Load other** option you can open a dialog and load another calibration file. However you cannot load a calibration that does not match the current camera system. If you need to do that you have to reprocess the calibration, see chapter “Reprocess calibration” on page 279.

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 327 and “Batch processing” on page 329.

The **Calibration results** are the same as shown after the calibration, see chapter “Calibration results” on page 271.

Calibration quality



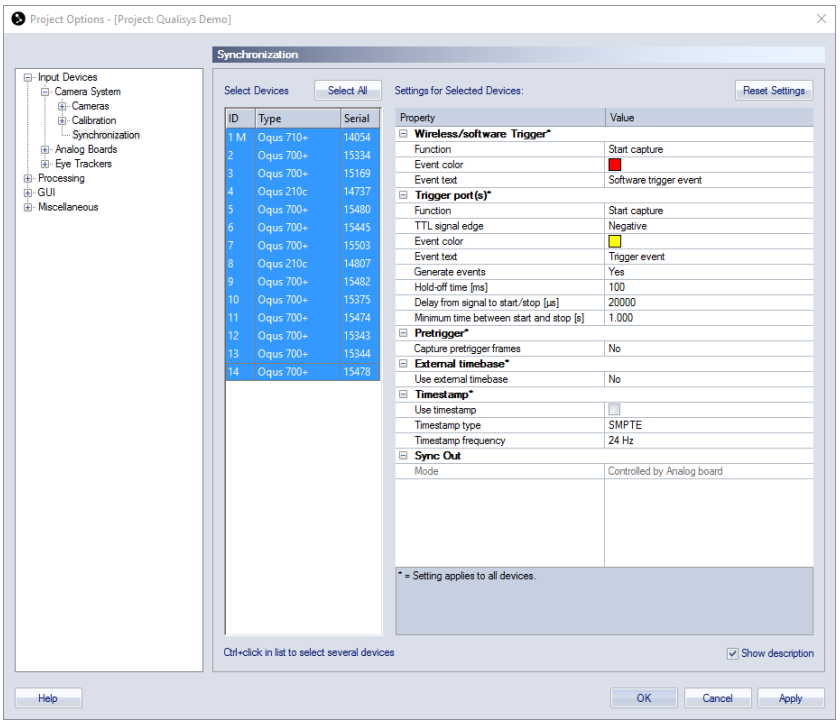
The **Calibration quality** page contains setting for how to detect if the cameras needs to be calibrated. The check can be performed in two different ways.

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 .

Synchronization



The **Synchronization** page contains all of the timing related settings. The following settings can be changed: **Wireless Trigger**, **Trigger port(s)**, **Pretrigger**, **External timebase**, **Synchronization output** and **SMPTE**, see chapters below.

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.

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.

Note: When a Camera Sync Unit is included in the system, QTM only displays the Synchronization settings of the Camera Sync Unit. The synchronization settings of Oqus cameras are not displayed. The use of **Pretrigger** is only supported for Oqus systems without Camera Sync Unit.

Wireless trigger

Wireless Trigger*	
Function	Start capture
Event Color	
Event text	Software trigger event

The **Wireless trigger** settings contain all the settings for triggering the Qualisys system using a wireless trigger, the keyboard or RT client applications. Possible input:

Qualisys wireless trigger
Use the **Trig** button for starting and/or stopping captures and use the **Event** button for creating events

Keyboard
Use **space** for starting captures. The **space** key cannot be used for stopping captures or creating events. In addition, the **PageDown** key can be used for starting and stopping measurements, and the **PageUp** key for settings events.

Note: Alternatively, the keyboard shortcut **Ctrl+M** can be used for stopping an ongoing capture (independent of wireless trigger settings).

RT Client application
For example mobile apps and plugins for Matlab and Labview.

Warning: The wireless trigger or keyboard cannot be used to trigger external equipment (e.g. EMG devices). Simultaneous triggering of the camera system and external devices require the use of an external trigger connected to a trigger port.

The following settings are available:

Function
Select the wireless trigger function from the drop-down list.

Start and stop capture disabled
The Wireless trigger or keyboard is not used for starting or stopping captures.

Start capture
The start of a capture is delayed until a trigger event is received.

Stop capture
An incoming trigger event will stop the capture.

Start and stop capture
Both the start and stop of the capture is controlled by the trigger event.

Note: The trigger settings of the Wireless trigger and the trigger ports are complementary, e.g., when the function of multiple trigger ports and the wireless trigger is set to **Start capture**, any of these trigger events will start a capture.

Note: When choosing **Stop capture** or **Start and stop capture** the capture will also stop if the end of measurement time is reached, unless you are have checked the **Continuous capture** option in the **Start capture** dialog.

Note: The stopping behavior of RT client applications may deviate from the **Function** setting.

Event color
Color associated with this type of event. Click in the value field to pick a color.

Event text
Text label associated with this type of event. Click in the value field to edit.

Trigger ports

Trigger port(s)*	
Function	Start capture
TTL signal edge	Negative
Event Color	
Event text	Trigger event
Generate events	Yes
Hold-off time [ms]	100
Delay from signal to start/stop [µs]	20000
Minimum time between start and stop [s]	1.000

The **Trigger port(s)** settings contain all the settings for triggering the Qualisys system using one or more external trigger devices.

For Oqus systems the use of an external trigger device requires a splitter cable connected to the control port of one or more of the cameras. The Trigger port settings apply globally to the control ports of all cameras within the system.

For Miqus systems a Camera Sync Unit (CSU) is required, see chapter “Camera Sync Unit: front side” on page 418. The trigger port settings can be controlled individually for the two trigger inputs **Trig NO** and **Trig NC**.

The Trigger input on Oqus cameras and the Trig NO (normally open) input on the CSU can be used with the Qualisys trigger button. The Trig NC (normally closed) input on the CSU requires an active trigger device for producing a "high" level (5 Volt).

The following settings are available:

Function/Trig NO: Function/Trig NC: Function
Select the external trigger option from the drop-down list.

Start and stop capture disabled
The external trigger device is not used for starting or stopping captures. However, you can still use the external trigger to create events.

Start capture
The start of a capture is delayed until an external trigger event is received.

Stop capture
An incoming trigger event will stop the capture.

Start and stop capture
Both the start and stop of the capture is controlled by the trigger event. For this option a **Minimum time between start and stop** is specified, since the two signals otherwise could be confused, see below.

Note: The trigger settings of the Wireless trigger and the trigger ports are complementary, e.g., when the function of multiple trigger ports and the wireless trigger is set to **Start capture**, any of these trigger events will start a capture.

Note: When choosing **Stop capture** or **Start and stop capture** the capture will also stop if the end of measurement time is reached, unless you are have checked the **Continuous capture** option in the **Start capture** dialog.

Note: When choosing **Stop capture** or **Start and stop capture** events cannot be created with the external trigger.

TTL signal edge/Trig NO: TTL signal edge/Trig NC: TTL signal edge
Select the trigger edge to **Positive** (rising edge), **Negative** (falling edge) or **Any** edge (rising or falling). The setting applies to all start, stop and event signals arriving at this trigger port.

Note: The Qualisys trigger button connected to an Oqus Trig in connector or the CSU Trig NO input will produce a negative signal edge when pressing the button, and a positive signal edge when releasing the button. The button is not debounced though, so positive edges may be produced even when pressing the button.

Event color/Trig NO: Event color/Trig NC: Event color
Color associated with this type of event. Click in the value field to pick a color.

Event text/Trig NO: Event text/Trig NC: Event text
Text label associated with this type of event. Click in the value field to edit.

Generate events
Activate this option to generate events with the external trigger. This option applies to all trigger ports, and it is activated by default.

Hold-off time
Hold-off time from the last event until a new event can be generated with the external trigger. This helps to avoid the creation of spurious events due to contact bounces when pressing or releasing the button. The hold-off time is by default 100 ms (range 5–1000 ms).

Delay from signal to start/stop [µs]
Delay (in microseconds) from trigger start event to first camera frame. The delay must be larger than or equal to 20000 µs (20 ms).

Minimum time between start and stop [s]
Minimum time between a start and a stop signal from an external trigger device. This time interval applies to signals coming in at any of the trigger inputs (not including wireless/software trigger). The range is 0.5–1000 seconds.

Note: For details on how to connect an external trigger device see chapter “How to use external trigger” on page 475.

Event port (Camera Sync Unit)

Event port*	
Hold-off time [ms]	100
TTL signal edge	Negative
Event Color	
Event text	Miqus Sync Unit Event

The **Event port** settings contain all the settings for creating events using an external trigger device connected to the event input of the Camera Sync Unit.

The following settings are available:

TTL signal edge
Select the trigger edge to **Positive** (rising edge), **Negative** (falling edge) or **Any** edge (rising or falling).

Note: The Qualisys trigger button connected to the event input will produce a negative signal edge when pressing the button, and a positive signal edge when

releasing the button. The button is not debounced though, so positive edges may be produced even when pressing the button.

Event color
Color associated with this type of event. Click in the value field to pick a color.

Event text
Text label associated with this type of event. Click in the value field to edit.

Note: When using IRIG as external timebase source, the Event port settings are ignored, since the IRIG time code signal needs to be connected to the **Event** input on the Camera Sync Unit.

Pretrigger (Oqus)

Pretrigger*	
Capture pretrigger frames	Yes
Pretrigger time [s]	2.000
Pretrigger marker frames	200
Pretrigger Video frames	6
Trigger start event color	
Trigger start event text	Post trigger start event

For Oqus systems, it is possible to use a pretrigger mode, so that actions can be captured that occurred before a trigger start event. The pretrigger mode is activated by setting **Capture pretrigger frames** to **Yes**. The pretrigger settings apply globally to the system. In pretrigger mode the cameras continuously update their buffer memories with a predefined number of frames until a trigger start event occurs. After the trigger start event, the pretrigger frames will be sent to the measurement computer. The trigger start event can be generated by an external trigger device, a wireless or software trigger, or by pressing the space key on the keyboard.

The following settings are available:

Capture pretrigger frames
Set to **Yes** for using pretrigger mode

Pretrigger time [s]
Time to buffer data (in seconds) before the trigger start event.

Pretrigger marker frames
Number of marker frames to buffer before the trigger start event.

Pretrigger video frames
Number of video frames to buffer before the trigger start event.

Note: The number of video frames can differ between cameras dependent on their video capture rate, which can be set individually. If the number of pretrigger video frames is different when multiple cameras are selected, it will be indicated red.

Trigger start event color
Color used to label the trigger event

Trigger start event text
Text label for the trigger start event

Pretrigger mode is not supported by all external hardware. For using pretrigger in combination with the USB-2533 analog board the use of an external trigger is required, see chapter “Measurement with analog capture while using pretrigger (Oqus)” on page 475 for details on how to connect the external trigger device. For AMTI Gen5 or Optima pretrigger functionality is supported by QTM; no additional hardware

connections are required. The pretrigger option is not supported in combination with EMG devices and A/V devices.

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
Use nominal frequency	☑
Nominal frequency	50.000
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. There are several ways to synchronize camera frames to an external timebase. For more information on how to use external timebase, see chapter "How to use external timebase" on page 476 .

The external time base should be connected to the corresponding input connector of the Camera Sync Unit, see chapter "Camera Sync Unit: front side" on page 418.

When using an Oqus camera for synchronization the external timebase can either be connected via the **Sync in** connector of the splitter cable (see chapter "Control connections" on page A - 15), or the **Sync in/Video in** or **Trig in/SMPTE in** connector of the Oqus Sync Unit (see chapter "Oqus Sync Unit" on page 423). It is recommended that you connect the external timebase signal to the master camera.

Note: It is important to use a stable signal source to achieve the best possible synchronization.

Note: Qualisys video cameras can capture at a different frequency (a divisor) compared to the marker cameras, see chapter "Using Qualisys video with External timebase" on page 479.

The following settings are available:

Use external timebase
Activate the use of an external timebase with **Yes**.

Signal source (Camera Sync Unit)

A Camera Sync Unit can be synchronized with the following signals. Make sure that the signal source is connected to the corresponding input connector on the Camera Sync Unit.

SYNC
Use a TTL signal to synchronize the system.

SMPTE
Use a SMPTE signal to synchronize the system.

Note: The **Use SMPTE timestamp** option is activated automatically when selecting SMPTE as signal source.

GENLOCK
Use a video signal (black burst) to synchronize the system.

IRIG
Use an IRIG signal to synchronize the system.

Internal 100 Hz
The system is synchronized with an internal 100 Hz signal. This option will be automatically selected if **100 Hz continuous** has been selected as **Synchronization output mode** on one of the synchronization output ports, see chapter “Synchronization output” on page 162.

Signal source (Oqus)
Oqus cameras 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** connector of the splitter cable or the Oqus Sync Unit. This is the standard method of synchronizing.

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 the system. You need to use the Oqus Sync Unit to convert the signal, see chapter “Using Oqus sync unit for synchronization” on page 486.
Note: The **Use SMPTE timestamp** option is activated automatically when selecting SMPTE as signal source.

Video Sync
Use a video signal (black burst) to synchronize. You need to use the Oqus Sync Unit to convert the signal, see chapter “Using Oqus sync unit for synchronization” on page 486.

Internal 100 Hz
The system is synchronized with an internal 100 Hz signal. This option can only be selected if **100 Hz continuous** has been selected as **Synchronization output** on one of the cameras, see chapter “Synchronization output” on page 162.
Note: This option is needed for the Twin system frame synchronization option, in which case it is selected automatically.

Signal mode
Select the way to synchronize the camera frame to the signal.
Note: Not available for all choices of signal source.

Periodic
In this mode the system locks on to the signal of the external timebase. The capture frequency can be set as a multiplier/divisor. This is the recommended setting for periodic signals see chapter “Using External timebase for synchronization to a periodic TTL signal” on page 476.

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.

Note: 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.

Note: The minimum delay from signal to shutter opening is 19000 µs in non-periodic mode.

Frequency multiplier/divisor
Set the multiplier respectively divisor to multiply and divide the incoming sync signal.

Frequency tolerance in ppm of period time
Set the tolerance for jitter of external periodic signals in ppm of the period time. The period time of the external signal should not differ more than the specified tolerance for the system to lock on to the frequency.

Note: Not available for the **SMPTE** signal.

Use nominal frequency
Activate the use of nominal frequency. When activated QTM will calculate the capture frequency based on the nominal frequency of the external timebase so that the relationship is always correct. This is recommended so that you can easily compare the data to that of an external system. If you do not use the nominal frequency, QTM will export the capture frequency with one decimal. In that case, the capture frequency may slightly deviate from what you expect due to the difference in clock speed between the Qualisys system and the external signal generator.

Nominal frequency
Enter the nominal frequency of the external timebase signal.

Note: When using SMPTE as signal source the nominal frequency value is read from the **SMPTE frequency** setting under **Timestamp**.

TTL signal edge
Specify the signal edge used for synchronization, either negative (falling edge) or positive (rising edge).

Delay from signal to shutter opening
Specify the delay of camera exposure relative to the synchronization signal in microseconds. For the **Periodic** mode the delay is 0 by default and can be set to any value. For SMPTE as a signal source the minimum delay is 100 µs.

For the **Non-periodic** mode the default and minimum time is 19 ms. 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 a 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 never exceeds the maximum frequency of the cameras in the current mode. The frequency should be limited to 97.5% of the maximum frequency at full image size. For an overview of maximum frequencies per camera, see “Qualisys camera sensor specifications (marker mode)” on page A - 1.

Timestamp

Timestamp*	
Use timestamp	☐
Timestamp type	SMPTE
Timestamp frequency	30 Hz

QTM has the option to add timestamps to camera frames for synchronization with external signals or devices. The timestamp is displayed in the **Timeline control** bar see chapter “Timeline control bar” on page 59.

The following settings are available:

Use timestamp
Check to add a timestamp to the camera frames.

Timestamp type
Select type of timestamp from the drop down menu. The options are:

SMPTE
Time code used for audio and video synchronization. This requires an Oqus or Camera Sync Unit to convert the SMPTE signal. For more information about using SMPTE, see chapters “Using Oqus sync unit for synchronization” on page 486 and “Using SMPTE for synchronization with audio recordings” on page 488.

Note: The supported SMPTE frequencies are 24 Hz, 25 Hz and 30 Hz, without dropped frames.

IRIG
Time code standards by the Inter-Range Instrumentation Group. This requires a Camera Sync Unit. The IRIG standards currently supported are IRIG B (1 Hz) and IRIG A (10 Hz).

Note: The IRIG signal should be connected to the Event input port on the Camera Sync Unit. The IRIG signal should be IRIG A or B, DCLS type.

Camera time
Time of the exposure in seconds.nanoseconds. When used without external clock master, the reference time is the time at which the master camera was started. When using PTP synchronization with a GPS-based external clock master the reference time is 1 January 1970. For more information about the use of an external clock master, see chapter “How to use PTP sync with an external clock master (Camera Sync Unit)” on page 480.

Timestamp frequency
Select one of the supported SMPTE or IRIG frequencies from the drop down menu.

Synchronization output

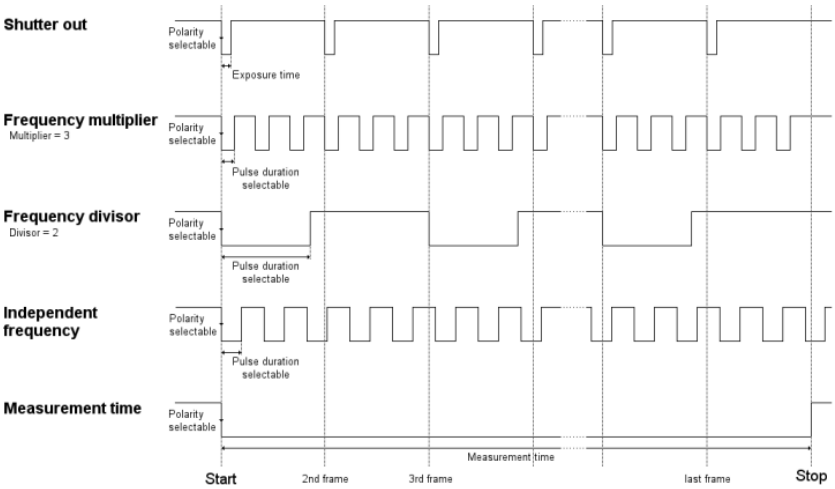
The Camera Sync Unit (CSU) has three synchronization outputs, **Out 1**, **Out 2** and **Measurement time**. The outputs **Out 1** and **Out 2** can be configured controlled to get a customized synchronization output.

For Oqus systems, each camera has a synchronization output, which can be accessed via a splitter cable connected to the control port. The **Synchronization output** can be controlled to get a customized synchronization output from the cameras. Additional modes for the Synchronization output for Oqus cameras are **Camera frequency – Shutter out**, **Measurement time** and **Wired synchronization of active markers**. The settings apply to individual cameras so that there can be different synchronization outputs within the same system. Select the cameras in the device list to the left. The synchronization signal is sent as soon as the camera is measuring, i.e. both during preview and capture, with the exception of **measurement time**, which is only sent during capture, and **100 Hz continuous**, which is always sent. For a description of how to use the sync output see chapter "Using Sync out for synchronization" on page 484.

Note: When a Camera Sync Unit is included in the system, QTM only displays the Synchronization settings of the Camera Sync Unit. The synchronization settings of Oqus cameras are not displayed.

Note: If you want to use a sync output signal that is faster than 5000 Hz with an Oqus system, you must 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. The image below is a description of the different sync output settings.



Camera frequency - Shutter out (Oqus)

Synchronization output	
Mode	Camera frequency - Shutter out
TTL signal polarity	Negative

This is the default setting for the Oqus cameras. 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 high until the first frame when it will go low.

Frequency multiplier/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 **Sync output** properties (**Frequency [Hz]**, **Period time [µs]** and **Pulse duration [µs]**) are shown below the setting both for marker mode and video mode (if applicable), because the marker and video capture rates can be different. The **Sync output frequency** will change if 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.

Note: The video capture rate is individual per camera, which is indicated by a red number if it differs. Select only the camera that Sync out is connected to find out the frequency in video mode.

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 mode and (if applicable) 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.

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 **Period time [µs]** will then update to its corresponding value.

The **Duty cycle** setting for controls how long the pulse will be in percent of the current **Period time**. The actual **Pulse duration** (in µs) is displayed below.

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 pulse will always be synchronized with the camera capture. The following pulses will be dependent on the relation between the camera capture rate and the **Output frequency**.

Measurement time (Oqus)

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.

Note: For the Camera Sync Unit, measurement time is not available as a mode for the **Out 1** and **Out 2** ports. Instead, the it has a separate **Measurement time** output with the same options.

100 Hz continuous

Use this mode to output a continuous 100 Hz TTL signal. It is sent even when the system is not measuring so that external equipment can lock on to this signal. In Oqus systems, only one camera can have this option activated.

Note: When **External timebase** is activated it is automatically changed to the **Internal 100 Hz** option. In this scenario, the device which is set to **100 Hz continuous** output will be used as a timebase.

Note: This option is required for the Twin system frame synchronization option, in which case it is selected automatically.

Note: The **100 Hz continuous** output is not available on Oqus 1, 3 and 5 series cameras.

Wired synchronization of active markers (Oqus)

Use this mode to synchronize the active markers via a wire connected from Sync out on the camera to the Sync in connector on the driver. The signal is pulsed so it cannot be used for any other synchronization.

Note: The mode is only available if you are measuring with active markers.

Controlled by Analog board

The camera synchronization output is controlled by the settings for an analog board, see chapter “Sample rate” on page 167.

Measurement time (Camera Sync Unit)

Measurement time	
TTL signal polarity	Negative

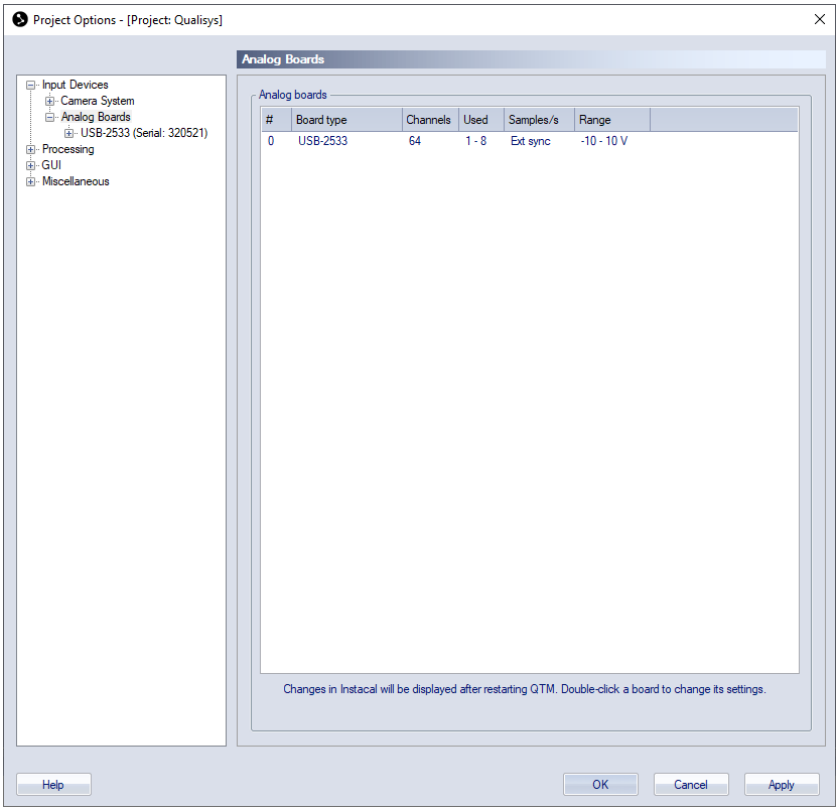
Use this the **Meas. time** output on the Camera Sync Unit get a pulse that last for the whole measurement time. The output will go low/high at the start of a capture and not go high/low until the end of the last frame.

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.

Analog boards

The analog boards that have been 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 450. For instructions on how to install an analog board, see “Appendix C: Analog input/output hardware”.

Note: If more than one board is installed make sure that the synchronization cable is connected to all of the analog boards.



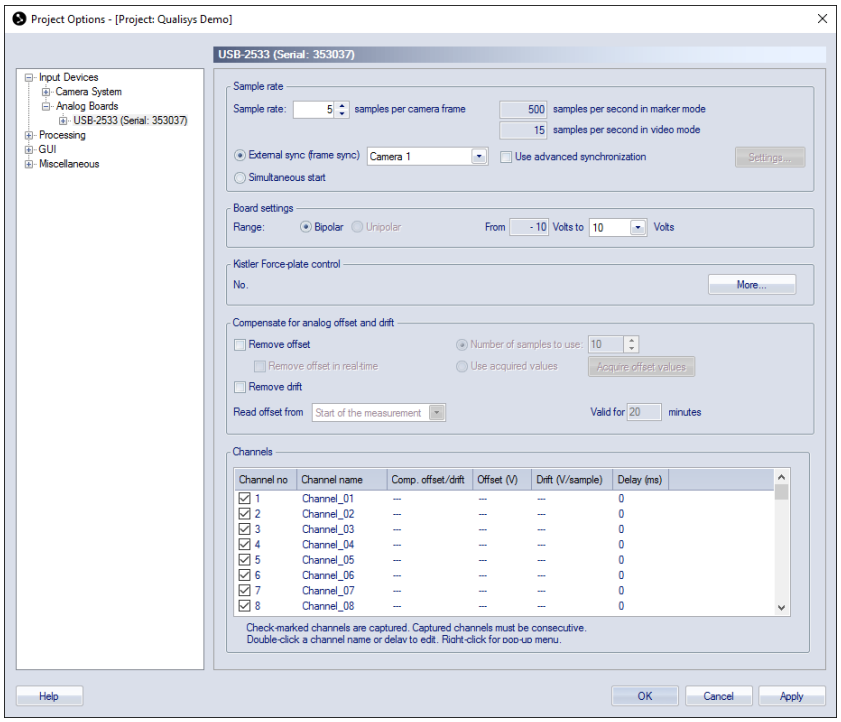
The list of the analog boards contains six columns: **#**, **Board type**, **Channels**, **Used**, **Samples/s** and **Range**. The list shows an overview of analog boards that are checked as **Enabled** on the **Input Devices** page.

The # column contains the number of the analog board in Instacal. The **Board type** column shows the type of the analog board, for a list of boards that are compatible with QTM see “Appendix C: Analog input/output hardware”.

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 settings” below.

Analog board settings

The page of each analog board contains the following settings: **Sample rate**, **Board settings**, **Force-plate control**, **Compensate for analog offset and drift** and **Channels**. For information on the settings see respective chapters below.



Sample rate

The **Sample rate** options control the frequency of the analog board. The sample rate is most often set in a multiple of the camera frequency, because it is easier to handle the data that way and this is for example a requirement for the C3D file. The default setting is one sample per camera frame, i.e. the same sample rate as for the camera system. What sample rate to use really depends on what you want to measure, for example for force data it is often enough with 500 to 1000 Hz.

The actual sample rate, in samples per second, for the analog board is shown in the text boxes to the right. When the **External sync** option is used with an Oqus camera as synchronization source, the sample rate is displayed both for marker mode and video mode . For all other options only the sample rate in marker mode is displayed.

Note: The minimum analog sample rate is 20 Hz, which is a limitation set by the analog driver.

There are then two options for synchronizing the analog board to the camera system: **External sync** and **Trigger start**.

> **External sync (frame sync) with**(default if available)
> The analog board is frame synchronized so that the drift between the camera system and the analog data is eliminated. This is the recommended setting for a Qualisys system with a USB-2533 analog board. You can use any frequency for the analog board. For instructions on how to connect the sync out signal from an Oqus camera or a Camera Sync Unit to the analog board see chapter “Connection of analog board” on page 450.
>
> Note: This option is only available for the analog boards USB-2533 and USB-1616FS.
>
> There are two options for the **External Sync**:
>
> > **Camera selection**
> > 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 will be changed on the individual camera. The specified camera will automatically be set to a multiple of the capture rate and the **Multiplier** setting is automatically updated with the **Sample rate** option.

The frequency can be different on separate boards, but you then need to connect the boards to different cameras. For USB-1616FS the sample rate is always controlled by the master board.

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 **Synchronization** page will however be the same, so you can calculate the analog frequency if necessary.

Note: QTM will detect if there is a conflict with the Point Grey camera synchronization, because a Oqus camera cannot be used both to synchronize Point Grey cameras and analog boards.

Use advanced synchronization
When the **Use advanced synchronization** option is activated the analog frequency is controlled directly from the **Synchronization** page. This means that the **Sample rate** option is disabled and you have to set the frequency directly. The advantage is that you can use the divisor and independent frequency modes so that you can in fact get any possible frequency.

Settings
To change the sync output frequency click on **Settings** to go directly to the **Synchronization** settings for the selected camera. There you can use any option for **Synchronization output** that you like, see chapter "Synchronization output" on page 162.

Simultaneous start
The analog board is just started on the first frame of the camera system, which means that there can be a small drift between the cameras and analog data.

The sample rate can be different on separate boards.

Board settings

Under the **Board settings** heading on the page for the analog board, there are settings for the range of the analog board.

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. If you have the Kistler 5695 DAQ this is where the number of force plates are shown. 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 170.

Remove offset
Removes an offset from the analog data. This setting can be used to remove the voltage level of a unloaded force plate. The offset compensation can either be calculated from the measurement or from an acquired list. The offset can be removed in RT/preview and in a file. When removed in a file the offset value cannot be changed only turned on and off, so it is important to set the correct settings before capturing the 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 from the start of the measurement.

Calculate offset from the measurement
Use the **Remove offset** from and **Number of samples** to use options to define the analog samples that are used to calculate the offset. Activate offset compensation in real-time with **Remove offset in real-time**.

Remove offset from
Set which part of the measurement to use when calculating the offset in the measurement. The default is **Start of the measurement**, but it can be changed to use the **End of the measurement**if there is noise at the beginning of the measurements.
Note: For this offset setting it is important that the samples used in the 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 calculated offset is higher than 5 V QTM will give you a warning, see chapter “Analog offset warning” on page 451. However you can always remove the offset compensation manually in the file from the **Data info** window, see chapter “Analog data information” on page 91.

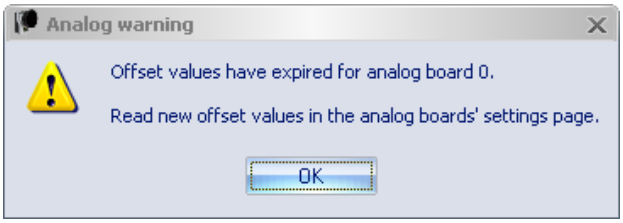
Number of samples to use
The offset is defined as an average of the number of samples that are defined in the **Number of samples to use** option.

Remove offset in real-time
Activate the **Remove offset in real-time** option to remove the analog offset during RT/preview as well. Only activate this if you are using the RT data. The offset will always be calculated from the beginning of the preview so it is important that this analog data is correct. 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 from the file.

Acquire list of offset values
The **Use values in list** below option can be used if you have a static offset that you can acquire during the real-time.

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 you can use 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 **Number of sample to use** option. This slope is then deleted from the analog data. This setting can be used to remove an output drift of a force plate that slowly changes the voltage level during the measurement.
The drift compensation is only applied on a file and cannot be used with the options **Read offset from end of measurement** and **Use values in list below**. Because those implicates that the analog data is not correct at the beginning of the measurement.
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 91.

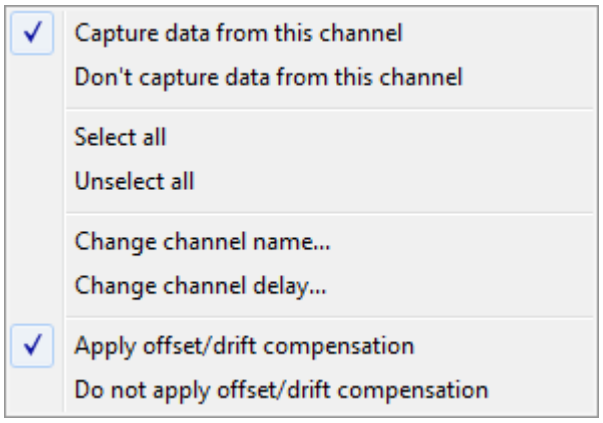
Channels

The channels of the analog boards are listed under the **Channels** heading. If you use the Kistler 5695A DAQ there are no settings for the channels, they are instead activated on the **Force plate control settings** page, see chapter “Force plate control list” on page 174.

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 **Channel name** column to rename the channel.

An individual delay in ms can be compensated for on each analog channel, for example it can be used with Delsys Trigno EMG that has a known delay of 48 ms. Double click on the **Delay (ms)** column to edit the delay of that channel. The delay is specified in ms, but the actual delay will be in the closest number of analog samples to that time. The compensation means that the analog samples on the channel is translated to match when it actually happened, for example with a 48 ms delay a signal that comes at 1.000 s will be translated to 0.952 s.
Note: The delay is not applied in real-time processing.

To edit the other settings right-click for the menu below.



Capture data from this channel
Use this channel, the same as selecting the checkbox.
Note: This option applies to all selected channels, i.e. the line is colored blue.

Don't capture data from this channel
Do not use this channel, the same as deselecting the checkbox.
Note: This option applies to all selected channels, i.e. that line is colored blue.

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.

Change channel delay...
Open a dialog to change the channel delay.

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.
Note: This option applies to all selected channels, i.e. that line is colored 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.
Note: This option applies to all selected channels, i.e. that line is colored 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.

Channel no	Channel name	Comp. offset/drift	Offset (V)	Drift (V/sample)
☑ 1	Channel_01	---	---	---

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.

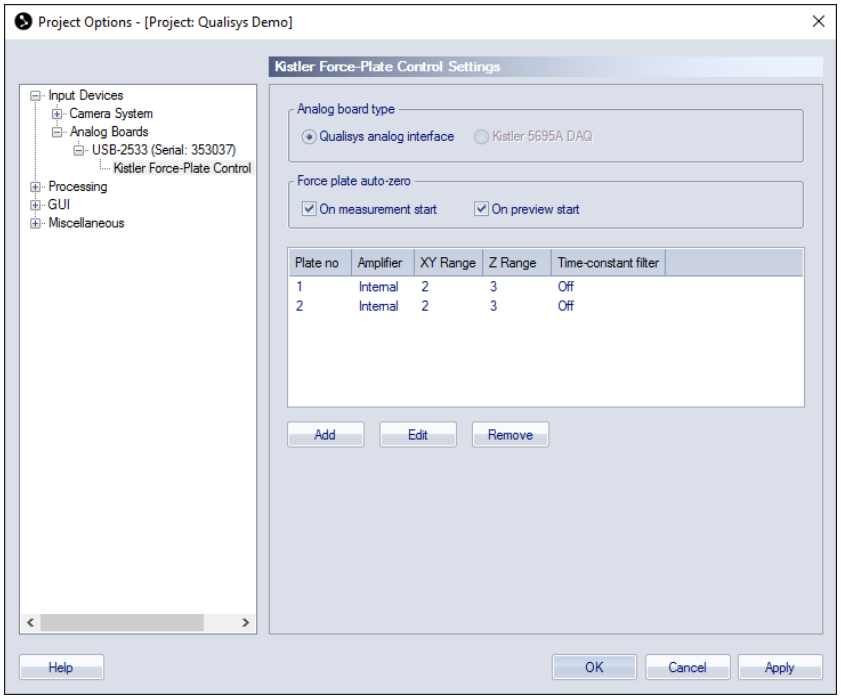
Channel no	Channel name	Comp. offset/drift	Offset (V)	Drift (V/sample)
☑ 1	Channel_01	Yes	-0.441	---

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 contain settings for the control of the force plate.

Note: These settings are only valid for the Kistler force plates. The Kistler force plates can, however, be externally controlled regarding Operate/Reset and charge amplifier range settings.

Analog board type

The **Analog board type** options makes it possible to select between the standard **Qualisys analog interface** and **Kistler 5695A DAQ**.

Qualisys analog interface
This is the default option that is used with all of the analog boards compatible with QTM.

Kistler 5695A DAQ
The option to select Kistler is only available with a USB-2533 board. This is the board used both in the 64 channel USB board and in the Kistler 5695A DAQ. Select the type of analog that is currently connected get the correct operation of the analog board. The force-plate control settings will change depending on the type that is selected, so it is important that you only select Kistler 5695A DAQ if that is actually connected.

Note: QTM does not support the Kistler 5691A DAQ, because it cannot be frame synchronized and the Instacal driver can malfunction so that the DAQ disappears from the computer.

Set the number of force plates and their ranges with the settings described in chapter “Force plate control list” on the next page.

Force plate auto-zero (Kistler)

Use the Force plate auto-zero settings to control when to auto-zero/reset the force plates connected to the analog board.

Note: This option is only available for Kistler plates connected to an analog board, not for the Kistler DAQ 5695.

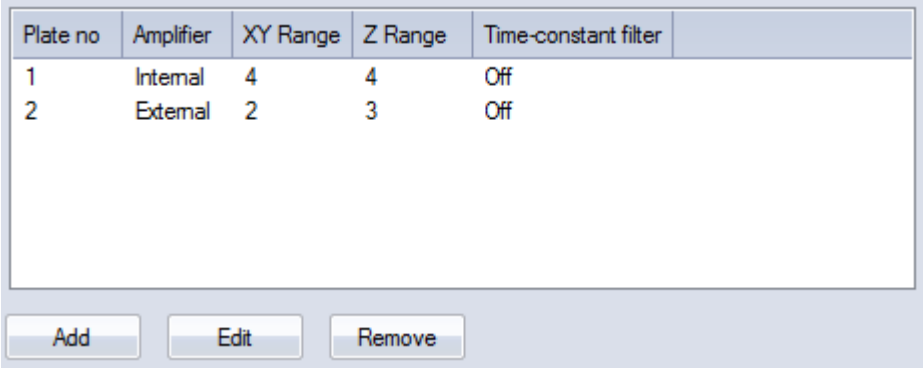
There are two options to control the auto-zeroing.

On measurement start
When enabled the force plates are auto-zeroed at two operations in QTM: just before the **Start capture** dialog is opened before a capture and in a batch capture just before QTM starts Waiting for next measurement/trigger.

On preview start
When enabled the force plates are auto-zeroed at two operations in QTM: new file and changing a setting in Project option during preview.

There is also an option to auto-zero the plates while QTM are in preview mode. Just right-click in a **Data info** window that displays force data and select **Zero all force plates**. It is important to use this zeroing at least every hour if both auto-zeroing options have been disabled.

Force plate control list



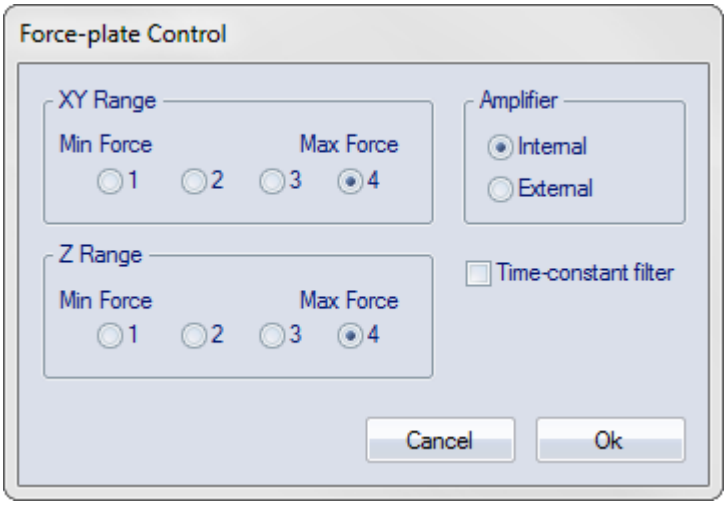
The list of force plate controls sets the controls and ranges of the connected Kistler force plates.To calculate the force data you must also specify the settings for the force plate, see chapter “Force plate” on page 211. The list and its functions are the same for all of the analog boards but it differs how many plates that can be connected to each board.

Analog board	No. force plates
USB-2533	4
Kistler 5695A DAQ	8

When connecting the force plates it is important to connect them in the same order to the analog channels as they are specified in the list. This is because the order is used to match the force plate numbers on the **Force plate** list on the **Force data** page, see chapter “Force data” on page 210. For more information on how to connect the force plates to the boards see chapters “Connecting Kistler force plates” on page 454 and “Connecting Kistler 5695A DAQ” on page 457.

A new Kistler plate is added with the **Add** option. The force plate control settings can then be edited or removed by selecting the plate and use respectively the **Edit** or **Remove** option. Adding a plate also means that a Operate/Reset signal will be sent to the force plate.

Note: For the Kistler 5695A DAQ the corresponding 8 analog channels are activated automatically when a force plate is added.



The **Force plate control** dialog contains the following settings:

Z Range/XY Range
Specify the range for the vertical (Z) and horizontal (XY) range of the force plate. The selected ranges determine the magnitude of the forces that can be measured with the force plate. The ranges of the amplifier can be different for the vertical and horizontal force components.

Note: The Kistler scaling factors will automatically be changed to the correct setting if the **Selected by forceplate control** options is used, see chapter “Kistler scaling factors” on page 214.

Amplifier
Select the type of amplifier used for the force plate, **Internal** or **External**.

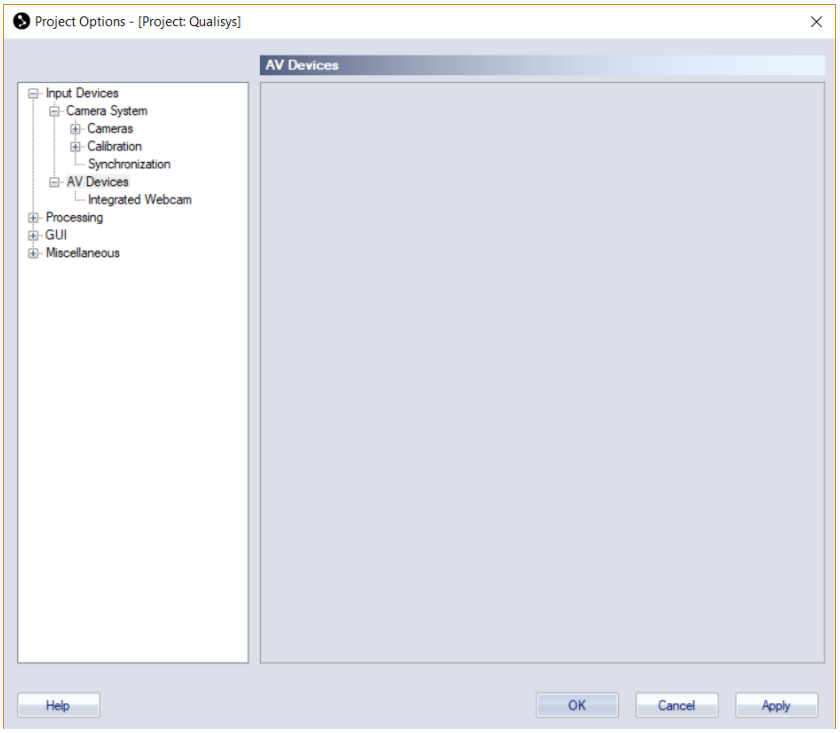
Time-constant filter
The **Time-constant** filter is useful for long measurements of static forces, for regular measurements the **Time-constant** filter should not be used.

Note : The filter cannot be used with the Kistler 5695A DAQ.

For further information about the range settings and the time-constant filter, see the manual for the Kistler force plate.

Note: With a breakout cable you can connect any analog signal to the Kistler DAQ. This can be useful if you have another type of force plate that you want to connect.

AV devices



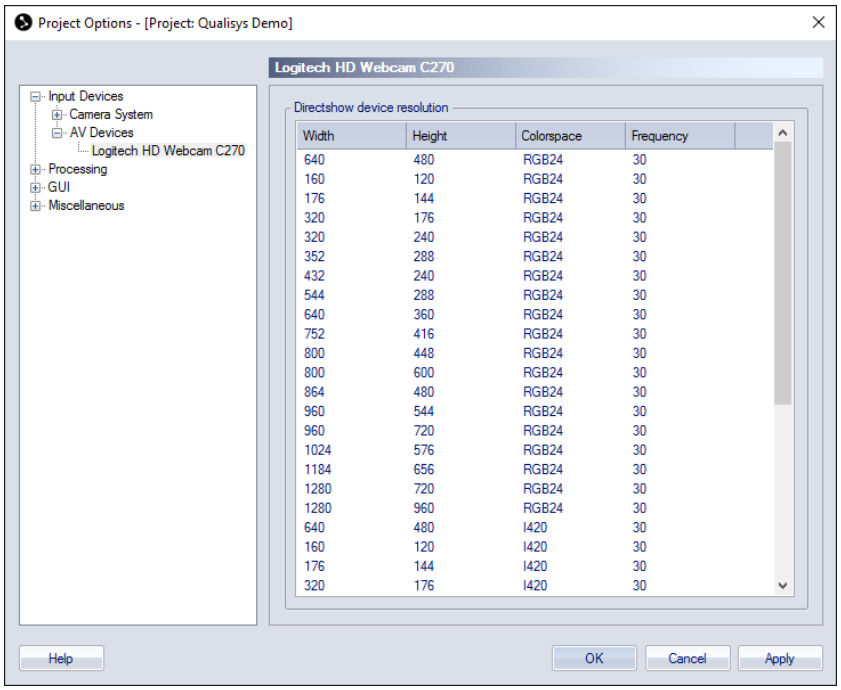
The external video devices that are selected under **Input Devices** are listed under the **AV Devices** node. This include video captured with Blackmagic Design cards and DV/webcam cameras, for more information on how to use them see chapter “Video capture with Blackmagic Design cards” on page 464 and “External video devices in 2D view” on page 36.

To use a video device go to the **Input Devices** page and select the check box in the **Enabled** column. 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.

When DirectShow video cameras are connected (for example web cameras), the **AV Devices** node can be expanded, giving access to a list of available video properties (resolution, color space and frequency).

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.

AV device settings



In the **AV devices properties** tab the video properties (resolution, color space and frequency) can be selected for DirectShow video cameras, for example web cameras.

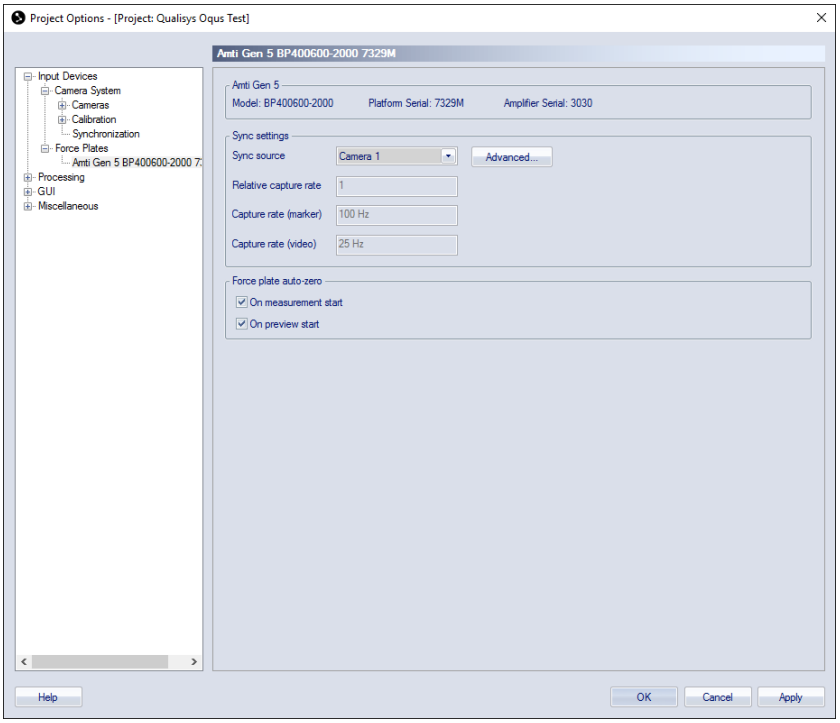
Force plates

The available AMTI force plates are listed in the settings tree to the left under the **Force plates** heading, see chapter “AMTI Gen5” below.

AMTI Gen5

The settings for AMTI Gen5, OPT-SC (Optima Signal Conditioner) and AccuGait Optimized are divided between the capture settings and the processing settings. The settings are the same for Gen5, OPT-SC and AccuGait Optimized so in the following descriptions we only refer to Gen5. On the **AMTI Gen5** page there are only settings for the capture. There is one page for each connected AMTI Gen5 amplifier, the name of the page will be the same as specified for the plate on the **Force data** page. The default name includes the model and serial number of the connected force plate.

For AMTI force plates with the new calibration chip the only processing setting that is needed is the location, see chapter “Force plate location” on page 224. If you have an old AMTI force plate where you have to input the calibration matrix manually then that needs to be added in the AMTI-Netforce program. QTM will then automatically read the settings file for that program. For a description on how to connect the cameras see chapter “Connecting AMTI Gen5/OPT-SC/AccuGait Optimized” on page 460.



The **Amti Gen5** heading contains information about the force plate that is connected to the amplifier. The information includes the **Model** and the **Serial** numbers.

The only setting under **Sync settings** heading that can be set on the AMTI Gen5 is the **Sync source** option, to decide which camera that the synchronization cable is connected to. Then click on **Advanced** to change the frequency of the force plate. You will go to the **Timing Advanced** page with just the Sync source camera selected. Change the **Synchronization output** setting to the desired frequency, see chapter “Synchronization output” on page 162. It is recommended to use the **Camera frequency multiplier** option to set the frequency, because for example when exporting to C3D then the force frequency must be a multiple of the marker capture rate.

Note: If there is an analog board in the system, that is also frame synchronized, then it is recommended to use the same camera as synchronization source for the analog board and the AMTI Gen5. Then you can use the **Sample rate** option on the **Analog board** page to control the frequency on both of the boards.

The following information is displayed on the page:

Relative capture rate
The sample factor compared with the marker capture rate, which is the same as the multiplier used for the Sync output.

Capture rate (marker)
The resulting AMTI Gen5 capture rate when using a camera in marker mode as sync source.

Capture rate (video)
The resulting AMTI Gen5 capture rate when using a camera in video mode as sync source.

Note: The buffer of the AMTI Gen5 is 16 samples. Therefore if the number of analog samples is not an multiple of 16 there will be some empty force plate samples at the end of the measurement.

Use the **Force plate auto-zero** settings to control when to auto-zero/reset the force plate. There are two options to control the auto-zeroing.

On measurement start
When enabled the force plates are auto-zeroed at two operations in QTM: just before the **Start capture** dialog is opened before a capture and in a batch capture just before QTM starts Waiting for next measurement/trigger.

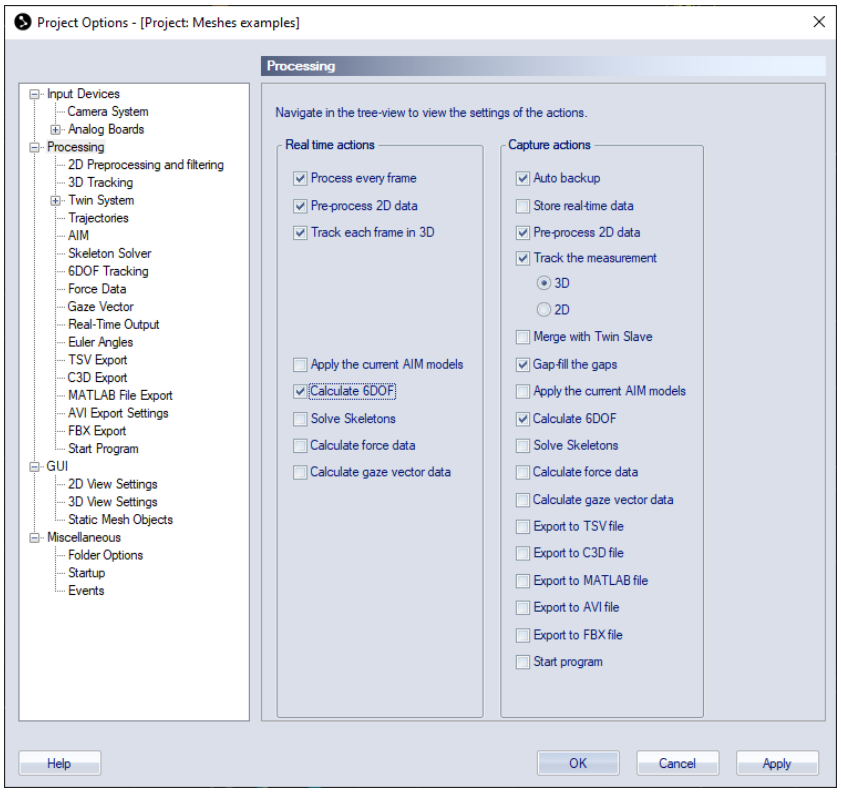
On preview start
When enabled the force plates are auto-zeroed at two operations in QTM: new file and changing a setting in Project option during preview.

Note: There is also an option to auto-zero the plates while QTM are in preview mode. Just right-click in a **Data info** window that displays force data and select **Zero all force plates**. It is important to use this zeroing at least every hour if both auto-zeroing options have been disabled.

EMGs

The available integrated EMGs are listed in the settings tree to the left under the **EMGs** heading, for each respective settings see chapters “Noraxon QTM settings” on page D - 9, “Mega QTM settings” on page D - 24 and “Delsys Trigno QTM settings” on page D - 32.

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 325 and "Real time in QTM" on page 313. For batch process a similar processing page is used see chapter "Batch processing" on page 329.

The actions that can be performed are as follows:

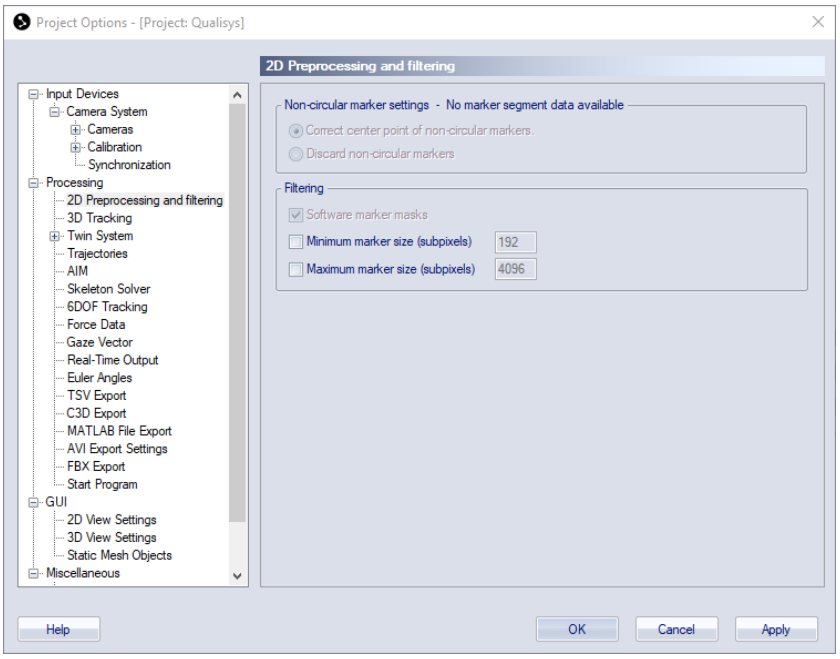
- **Auto backup** (only Capture action), see chapter "Auto backup" on page 284.
- **Process every frame** (only Real time action). For more information, see chapter "Outline of how to use real time" on page 315.
- **Store real-time data** (only Capture action), see chapter "Store real-time data" on page 284.
- **Pre-process 2D data**, see chapter "2D Preprocessing and filtering" on the next page.
- **Track each frame**
  - **3D**
  - **2D** (only Capture action)
- **Merge with Twin Slave** (only Capture action)
- **Gap-fill the gaps** (only Capture action)
- **Apply the current AIM model**
- **Calculate 6DOF**

- **Solve skeletons**
- **Calculate force data**
- **Calculate gaze vector data**
- **Export to TSV file** (only Capture action)
- **Export to C3D file** (only Capture action)
- **Export to Matlab file** (only Capture action)
- **Export to AVI file** (only Capture action)
- **Export to FBX file** (only Capture action)
- **Start program** (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.

2D Preprocessing and filtering



The **2D Preprocessing and filtering** page contains settings for the preprocessing of 2D data before 3D tracking. The available options are **Non-circular marker settings** and **Filtering**.

Non-circular marker settings (Oqus)

Use the **Non-circular marker** settings to define how to handle the segment data that is sent from the Oqus cameras. The cameras will only send segment data when the **Marker circularity filtering** option is enabled on the **Cameras** page, "Marker circularity filtering (Oqus)" on page 132. How many markers that have segment data depends on the **Circularity threshold** on that page.

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.

The options for handling non-circular markers with segment data is:

Correct center point of non-circular markers
With this option non-circular (partially hidden) markers with segment data are corrected, if possible, before they are used by the 3D tracker. The center point will be more exact for partially hidden markers than the marker calculated by the camera. The correction works best if the marker size is between 500 and 1500 subpixels. If the marker size is below 320 subpixels then the marker is not considered for correction since it is too small to find a good center point for.

Note: If the Marker circularity filtering is not enabled on the **Cameras** page, then the options are greyed out and the message **No marker segment data available** is shown. It is the same when reprocessing files that don't have any segment data.

Discard non-circular markers
With this option non-circular markers with segment data are discarded so that they are not used by the 3D tracker. This mode is faster than the correction options, so it can be used in some cases when the processing time is important.

For more information about how to use the Non-circular marker settings see chapter "How to use non-circular marker settings" on page 299 and "How to use circularity filter" on page 330.

Filtering

The **Filtering** option applies software marker masks and size filtering to the 2D data. The option is used on all of the cameras in the measurement. Markers that have been filtered are indicated by a rectangular frame in the camera view in the 2D view window.

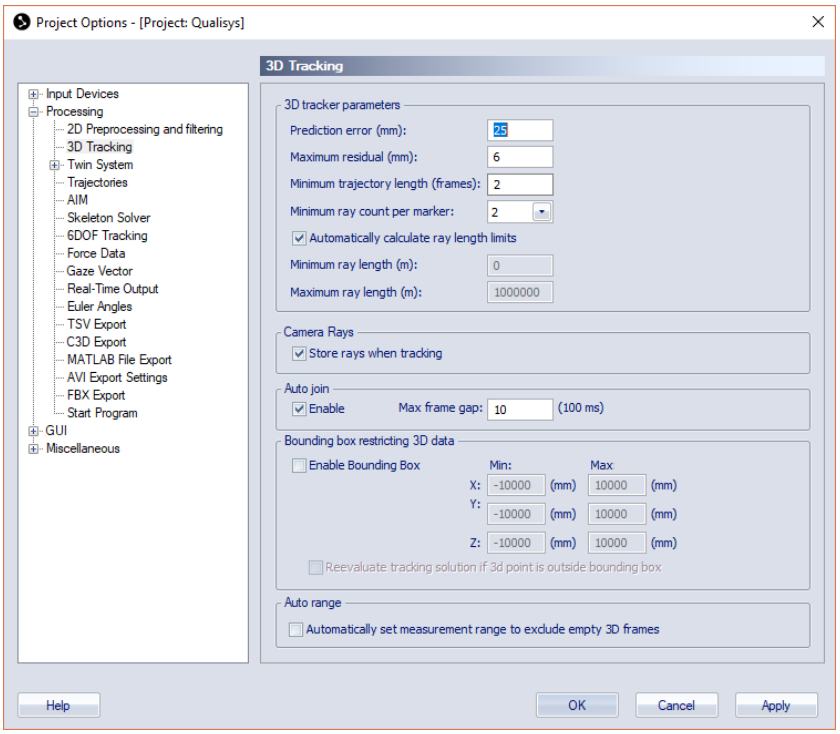
Software marker masks (only available when reprocessing a file)
Enable software marker masks in reprocessing, see chapter "How to use software marker masks" on page 331.

Minimum marker size
Set the minimum size for the 2D marker data in subpixel units to be used in the 3D tracking. For more information about marker size filtering see chapter "How to use marker size filter" on page 332.

Maximum marker size
Set the maximumsize for the 2D marker data in subpixel units to be used in the 3D tracking.

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 334.

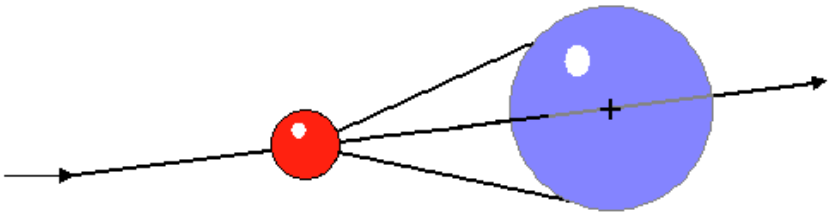


3D Tracker parameters

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. The contribution of individual cameras to the estimated 3D position of a marker are weighted so that rays closer to the center of the sensor are weighted higher than rays closer to the edges.

The behavior of the 3D tracker can be controlled by the following parameter: **Prediction error**, **Maximum residual**, **Minimum trajectory length**, **Minimum ray count per marker** and **Ray length limits**, see the chapters below for more information.

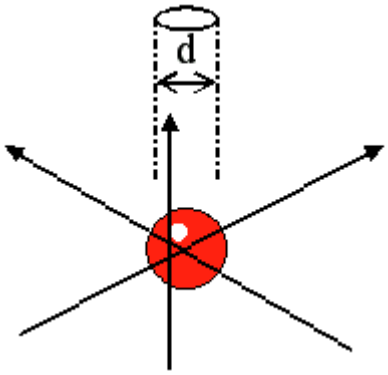
Prediction error



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 25 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 camera rays are considered to belong to that 3D point. The example above shows three camera 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 6 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.

Minimum ray count per marker

The **Minimum ray count per marker** parameter defines the minimum number of cameras required for the 3D tracking of a marker. The default value is 2, which imposes

no limitation to the 3D tracking.

Increasing this value adds constraints to the 3D tracker by requiring that at least the set number of cameras must be involved in the reconstruction of a 3D marker. This can be useful in situations in which tracking artifacts in the form of ghost markers occur. It is recommended to keep this value as low as possible since higher values may result in additional trajectory gaps and a decrease of the effective capture volume.

Ray length limits

The ray length limits represent the minimum and maximum distance between a marker and a camera to be used for for 3D tracking. The units are in meters.

When the option **Automatically calculate ray length limits** is checked, the ray length limits are automatically calculated based on the current calibration. When this option is unchecked, you can manually set the minimum and maximum ray lengths.

Rays

Camera tracking rays are a mapping between 2D data of the cameras and the 3D trajectories, which can be shown in the 3D view window, see chapter “Rays in 3D views” on page 52.

To show the rays, they to be stored during the 3D tracking stage of the processing. To store the rays, make sure that the option **Store** is enabled.

Note: When the **Store** option is enabled, the processing will take longer time and the file size of the QTM file will be larger.

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 matching number of milliseconds with the current frequency is displayed next to the option. 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 can be used to make the identification easier.

Note: Auto join only joins two trajectories so that they are one in the trajectory info windows. If you want to fill the gap between the two trajectories with data you need to use the gap-fill function, see chapter “Trajectories” on page 192.

Bounding box restricting 3D data

The bounding box is the volume in which the trajectories will be calculated. The use of a bounding box can be helpful to reduce the amount of unneeded data by discarding 3D data which falls outside the volume of interest. The bounding box is displayed as a white outlined box in the 3D view, the display is controlled on the **3D view settings** page.

When enabling the bounding box, the 3D data is restricted to the volume spanned up by the bounding box. The limits of the bounding box can be defined in the text edit fields as distances (in mm) in each direction to the origin of the global coordinate system. The default behavior is that 3D points that fall outside the bounding box are discarded, i.e. the option **Reevaluate tracking solution if 3D point is outside bounding box** is unchecked. By checking the option **Reevaluate tracking solution if 3D point is**

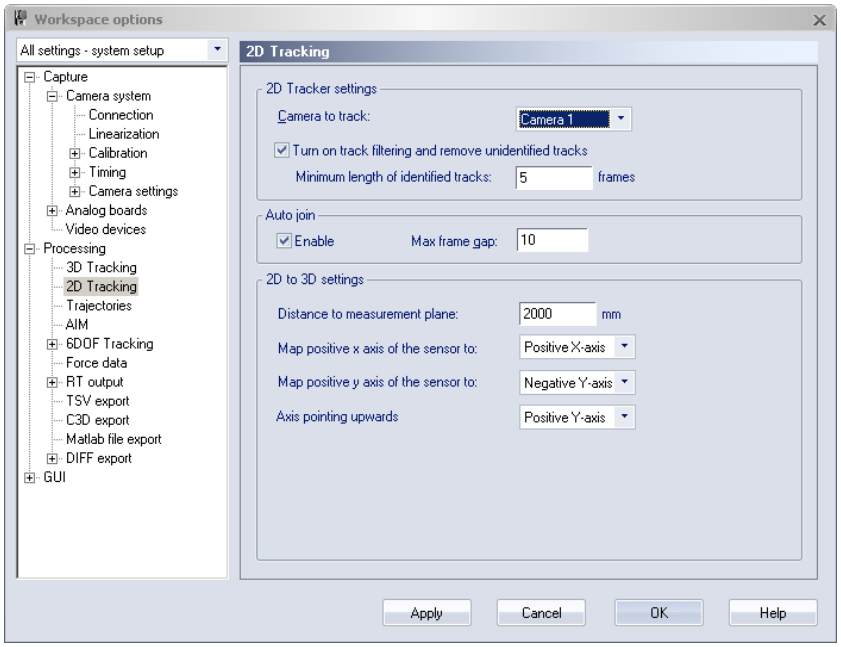
outside bounding box the 3D tracker will try to find alternative solutions for 3D points if the initial solution lies outside the bounding box. This can be useful for small systems, for example with two cameras.

Note: If you use a 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.

Auto range

The **Auto range** option automatically sets the measurement range so that it only includes frames that have 3D data. The empty frames are not deleted just placed outside the measurement range. Increase the measurement range on the Timeline control bar if you want to use more frames, see chapter “Timeline control bar” on page 59.

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 360.

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 matching number of milliseconds with the current frequency is displayed next to the option. 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 can be used to make the identification easier.

Note: Auto join only joins two trajectories so that they are one in the trajectory info windows. If you want to fill the gap between the two trajectories with data you need to use the gap-fill function, see chapter “Trajectories” on page 192.

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.

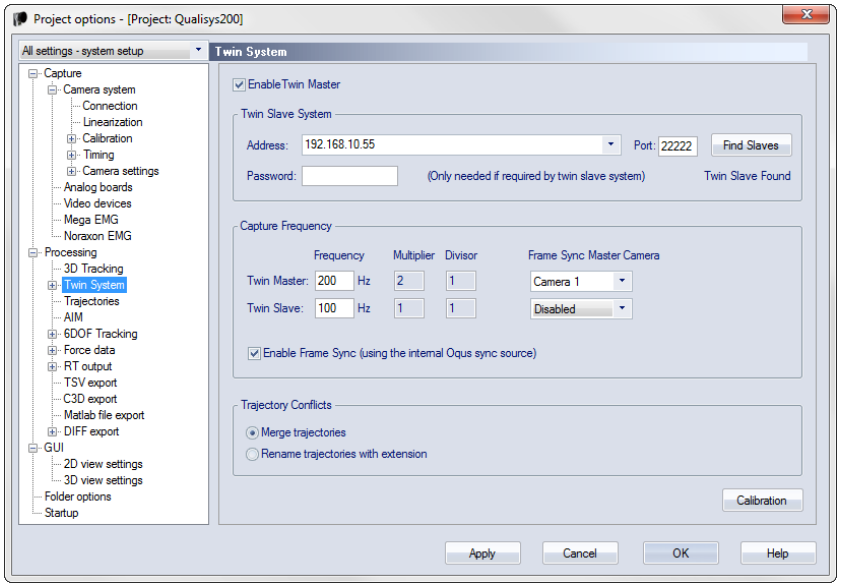
Map positive y axis of the sensor to

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.

Twin System



The settings on the **Twin System** page are used to enable and control the Twin system feature. This feature enables a Twin master system to control a Twin slave system on another computer. It can for example be used to combine an underwater system with one above water. For more information about the Twin system feature, see chapter “Twin systems” on page 301.

To enable the Twin master functionality select the **Enable Twin Master** option on the system that you want to use as Twin master. It is then important to set the other settings on the page for correct operation, see below.

Note: You do not need to change any settings related to Twin systems on the Twin slave system.

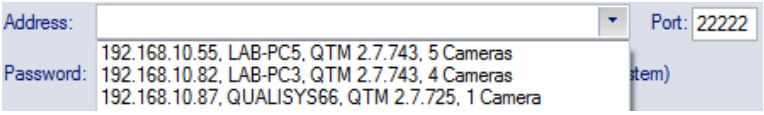
To enable the automatic transfer of the Twin slave file you must select the **Merge with Twin Slave** option on the **Processing** page, see chapter “Processing” on page 181.

The two systems must also be calibrated with a twin calibration, for information about the settings see chapter “Twin System Calibration” on page 191.

Twin Slave System

The **Twin Slave System** settings controls which slave is controlled by the Twin master system.

Use the **Find Slaves** button for QTM to search for any available QTM programs on other computers. QTM must be running on the other computer and connect via Ethernet (directly or via the LAN) to the Twin master computer. Only computers with a located camera system will show up in the list. Select the computer that will be the Twin slave from the list below the **Address** setting.



It is possible to enter an address manually in the **Address** setting if the **Find Slaves** option fails. The **Port** and the **Password** setting only needs to be changed if they have been modified in QTM of the Twin slave system.

Capture Frequency

The **Capture Frequency** settings control the frequency and synchronization of the Twin master and slave systems. The settings that are set for the two systems will override any other settings on the two systems. However, the timing settings will not change until you start a new measurement on the Twin master with **New** on the **File** menu. The following settings can be set:

Frequency
The frequency for the two systems must always be set. The Twin slave frequency cannot be higher than the Twin master. It is recommended to use the same frequency or a divisor of the Twin master frequency for the Twin slave, but it can be any frequency.

IMPORTANT: If you do not use the same frequencies for the two systems then the data of the slave system is interpolated. When you use a divisor of the twin master frequency, then the interpolated data is just filled between the actual 3D data. If the twin slave frequency is not a divisor of the twin master frequency, then QTM will interpolate all of the slave data so that it matches the twin master frequency.

Enable Frame Sync
When enabled the frame sync will control the **External timebase** settings of the two systems so that they are frame synchronized with each other. The use of Frame Sync is recommended. For information about how to connect the synchronization signal between the systems see chapter “How to use frame synchronized twin systems with separate volumes” on page 301.
You need to select the **Frame Sync Master Source** that will send the **100 Hz continuous** signal on its Sync out connection. The system with the Frame sync master source will be set to the **Internal 100 Hz** option for external timebase. The other system will be set to using the **Control port** or **SYNC** option. The **Multiplier** and **Divisor** options of the systems will be set to numbers displayed next to the **Frequency** setting.

Note: The **100 Hz continuous** output is not available on Oqus 1, 3 and 5 series cameras; the use of Frame sync requires at least one other type of Oqus camera or a Camera Sync Unit.

Frame Sync Master Source (only available with the Frame sync option)
Select the device that you want to use as frame sync master. This can be a sync output on the Camera Sync Unit or an Oqus camera. Devices with multiple sync out ports (e.g. Camera Sync Unit) will additionally let you specify which port to use (currently, only the Out 1 port on the Camera Sync Unit is supported for Twin). The Frame sync master source can be in either the Twin master or the Twin slave system.

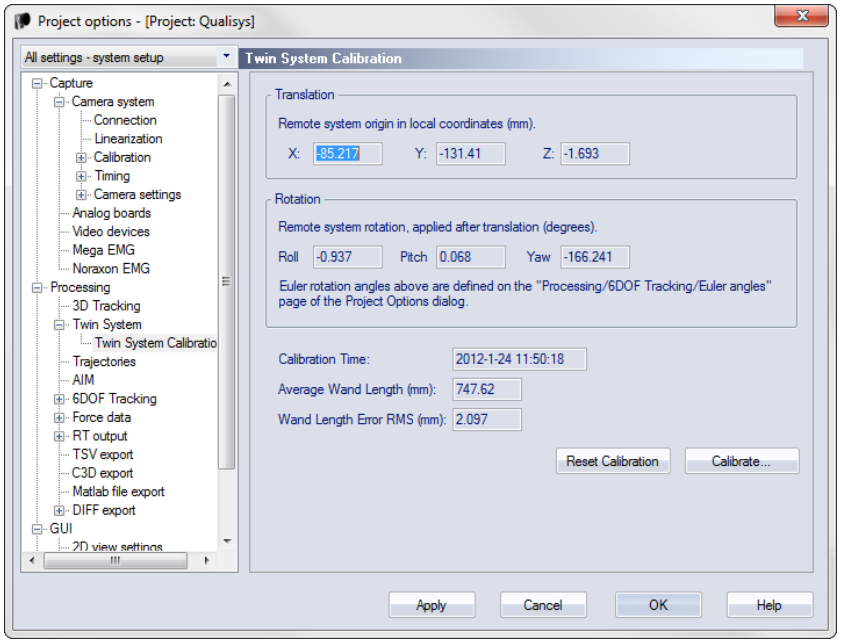
Trajectory Conflicts

If there are two labeled markers with the same names in the twin master and twin slave file, then QTM will have to know how to handle the trajectories. QTM can either **Merge trajectories** or **Rename trajectories with extension**. If the trajectories are merged then the twin slave trajectory will be added to the twin master trajectory and if there are any overlaps the twin master data is used. When renaming the trajectories then the extension _slave is added to all of the labeled twin slave trajectories.

Note: This option will only be used if the Twin slave file contains labeled trajectories. Most of the time it is recommended to not identify the data on the slave file and instead apply AIM to the merged data.

Note: If the volumes are completely separated there will be no overlaps in the merged data.

Twin System Calibration



The **Twin System Calibration** page contains settings for the twin calibration of the two systems and describes the **Translation** and **Rotation** that is used for transforming the twin slave data to the twin master coordinate system. For information on how to perform a twin system calibration see chapter “Performing a Twin calibration” on page 307.

The result of the last twin calibration is displayed below the transformations:

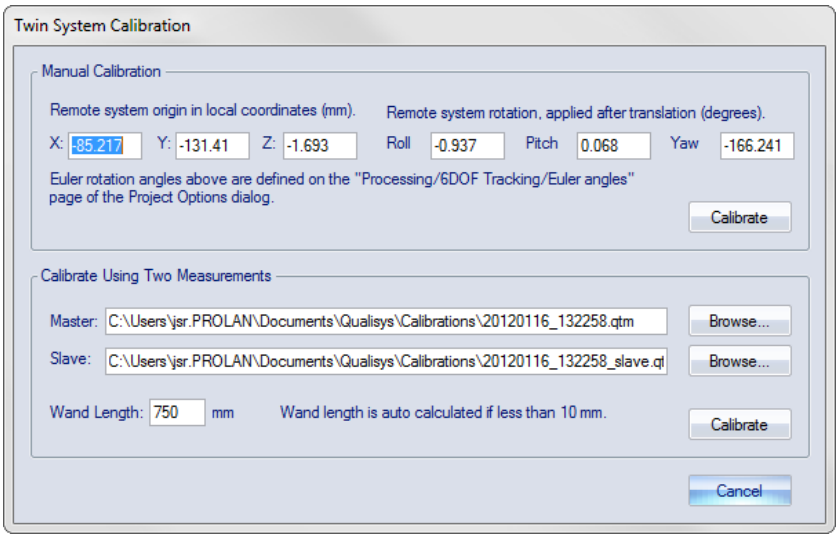
Calibration time
The processing time of the current twin calibration. Since it is only the processing time that is displayed you need to check which files that are used to be sure exactly what is used, see chapter “Twin System Calibration dialog” on the next page.

Average Wand Length (mm)
The average wand length of the current twin calibration.

Wand Length Error RMS (mm)
The wand length error of the current twin calibration.

Click on **Reset Calibration** if you want to zero the twin calibration data. Click on **Calibrate** to change the twin calibration, see chapter “Twin System Calibration dialog” on the next page.

Twin System Calibration dialog



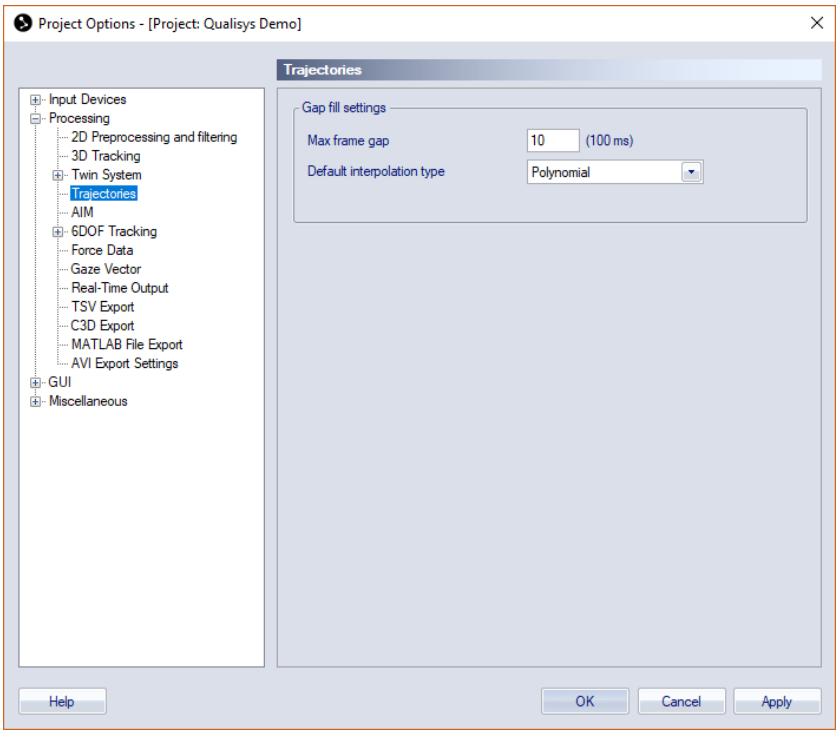
The **Twin system calibration** dialog is opened with the **Calibrate** button on the **Twin system calibration** page, see chapter “Twin System Calibration” on the previous page. From the dialog you can control and change the current twin calibration. It can either be changed by updating the measurements used for twin calibration or manually.

Manual calibration
Enter the translation and position of the twin calibration. If you have made a twin calibration from two files then it is the result that is displayed. However, you can also enter the numbers manually, but it is not recommended if you want the best accuracy. Click on **Calibrate** to update the twin calibration with the manually entered data.

Calibrate using two measurements
These settings displays the current files used for the twin calibration. You can click **Browse** to change the files or change the **Wand length**. Click on **Calibrate** to update the twin calibration.

Trajectories

The **Trajectories** page contains the setting **Gap Fill Settings**.



Gap Fill Settings

Under the **Gap Fill Settings** heading there are options for the gap fill functionality.

Set the **Max frame gap** option to select how many frames that can be gap filled without preview. The trajectories will only be gap filled if the gap between the two parts is less than the **Max frame gap**. The matching number of milliseconds with the current frequency is displayed next to option.

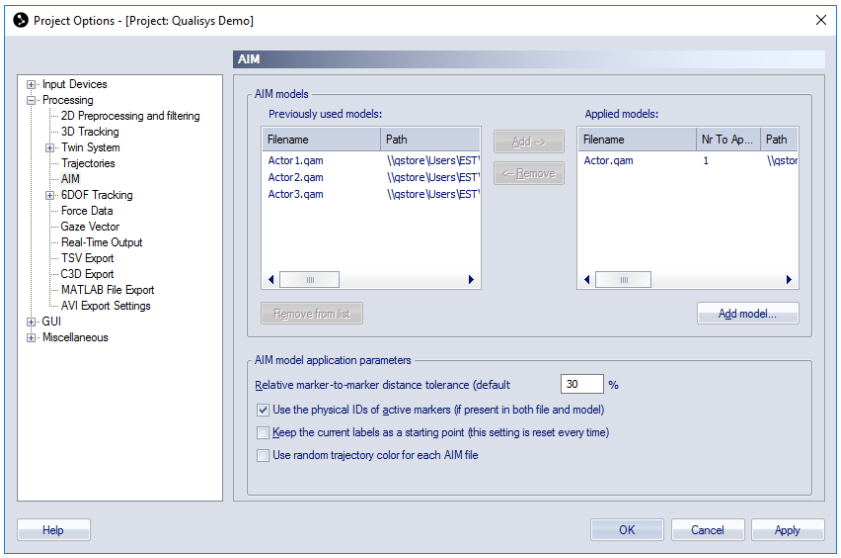
Select the **Default interpolation type** to decide whether **Polynomial** or **Linear** gap fill is used by default. The **Polynomial** gap fill uses two frames before and after the gap to calculate a third degree polynomial. The **Linear** gap fill calculates the line between the start and end frame of the gap.

The gap fill function is applied to trajectories either via the **Trajectory info window** menu or as a processing step. When gap fill is performed as a processing step gap fill is tried on every trajectory.

If gap fill is applied from the **Trajectory info window** menu, it is only the selected trajectories that are gap filled and the setting is retrieved from the **Max frame gap** setting saved in the file. To change the gap settings you need to reprocess the file with the new gap fill settings under the **Trajectories** processing options.

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 364.



AIM models

Under the **AIM models** heading there are two lists with AIM models. The **Applied models** list are the models that are currently in used by QTM. There can be several models in this list and then QTM will try to apply them all. Remember to **Remove** (moves the model to the **Previously used models** list) the unnecessary models from this list otherwise the AIM application might fail. Add a saved model to the list with **Add model**. If you want to apply the same AIM model to multiple actors, you can set the number of actors in the **Nr To Apply** column.

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 from 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.

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.

Use the physical IDs of active markers (if present in both file and model)
This setting only applies to files with active and passive markers in the same file. When activated AIM will use the active marker IDs to match the markers with the AIM model, which means that you will aid the AIM model when identifying the rest of the markers. The AIM model must have been created with active markers placed on the correct positions in the AIM model.

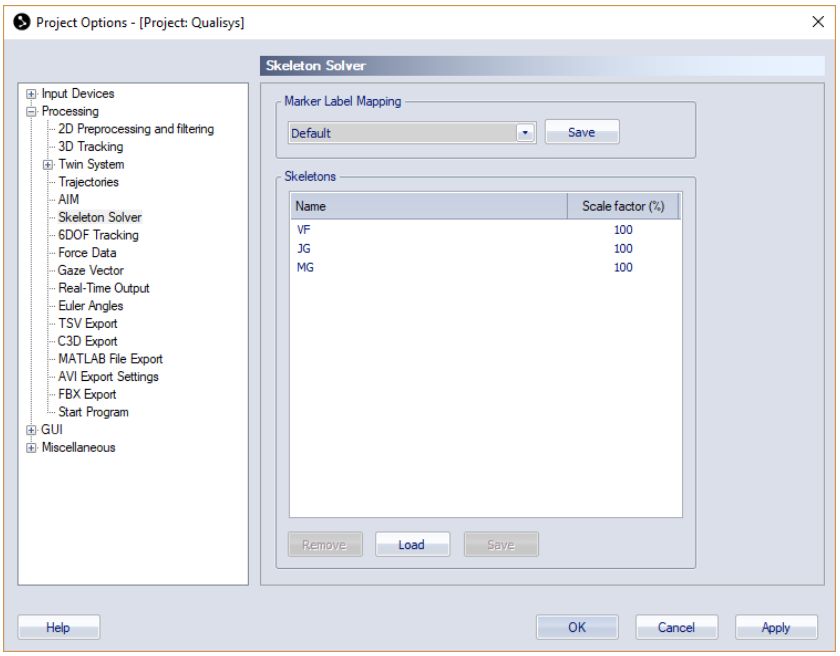
Keep the current labels as a starting point (this setting is reset every time)
Use this setting to help AIM with the identification in a file. When activated AIM will keep the identified trajectories as they are and try to find a solution for the rest of the trajectories that matches these.

Note: The setting is inactivated after each time it is used, because to be able to use it you must manually identify markers before applying AIM.

Use random trajectory color for each AIM file
Use this option to assign a random color to the trajectories for each applied

AIM model. This can be helpful to distinguish trajectories when using the same AIM model multiple times.

Skeleton solver



The Skeleton Solver page is used to manage the skeleton definitions used in the project. For more information on skeleton tracking, see chapter “Tracking of skeletons” on page 350.

Marker Label Mapping
Mapping of markers associated with a skeleton. See chapter “Skeleton marker label mapping” on page 353 for how use a custom marker mapping.

Skeletons
List of skeletons loaded in the project. The column are:

Name
Name of the skeleton, corresponding to the used prefix of the trajectory labels.

Scale factor (%)
Scale factor of the skeleton in percent used for streaming the data and FBX export. The scale factor is applied inversely. For example, for a scaling factor larger than 100% the skeleton data is scaled down. For more information about the use of the scale factor, see chapter “Scale factor” on page 359.

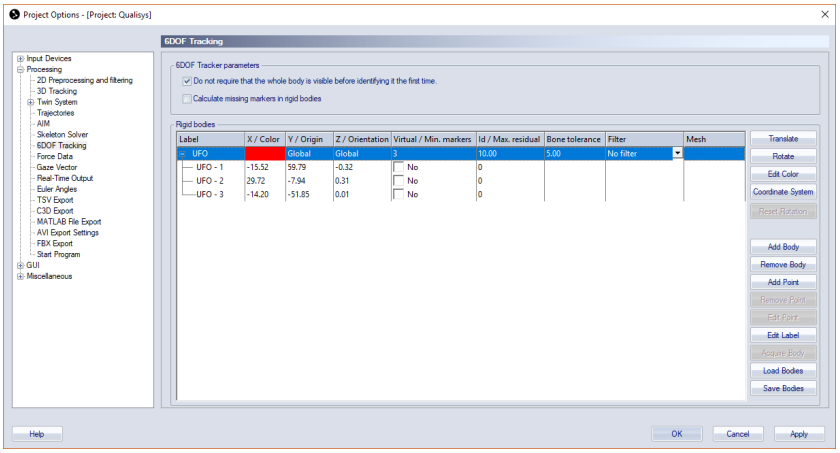
The skeleton list can be managed using the following buttons.

Remove: Remove the selected skeleton.

Load: Load skeleton from QTM skeleton file.

Save: Save selected skeleton to QTM skeleton file. If you select multiple skeletons, each skeleton will be saved in a separate file.

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 of rigid bodies” on page 336. For description of the **Rigid bodies** list see chapter “Rigid bodies” below.

It is also possible to export the 6DOF tracking output to another computer and as an analog signal, see chapter “Real-Time output” on page 227 respectively chapter “6DOF analog export” on page 228.

6DOF Tracker parameters

Specify the global tracker parameter for the 6DOF tracking under the **6DOF Tracker parameters** heading. More 6DOF tracker parameters, in particular **Min. markers**, **Max. residual** and **Bone tolerance**, are available for the individual rigid body definitions, see chapter “Rigid bodies” below.

Do not require that the whole body is visible before identifying it the first time
Enable this option to remove the requirement that all markers of the rigid body must be visible to identify the body the first time. This can help the 6DOF tracker to find a solution in case only a part of the body is visible.

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 343.

For information on 6DOF tracking see chapter “6DOF tracking of rigid bodies” on page 336.

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 the following columns. In case a column refers to separate items for the rigid body definition or its points, respectively, this is indicated by the / separator.

Rigid bodies:

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.

Origin and **Orientation**.
The coordinate system that the 6DOF body data refer to is displayed in the **Y / Origin** and **Z / 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 202.

Min. markers
Specify the minimum number of markers required for 6DOF tracking of the rigid body.

Max. residual
Specify the maximum residual accepted for 6DOF tracking of the rigid body.

Bone tolerance
The **Bone 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 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 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 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.

Filter
Select the filter for smoothing 6DOF data. The default is **No filter**. The filter is applied both in real time and in a capture. For more information about smoothing 6DOF data, see chapter “Smoothing 6DOF data” on page 204.

Mesh
Object file of 3D mesh associated with rigid body. Double-click on the mesh setting of the rigid body to open the **Mesh Settings** dialog, see chapter “Rigid body Mesh Settings dialog” on page 205.

Points:

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 343.

Id
The ID of the trajectory in case the point is associated with a short range active marker.

The options that are used to edit the rigid bodies or their points are described below:

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 the facing 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 200.

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 202.

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.

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.

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.

Edit label
Edit the selected label (rigid body or point).

Acquire body
Acquire the rigid body definition from the current marker positions in RT/preview mode, see chapter “Acquire body” on page 203.

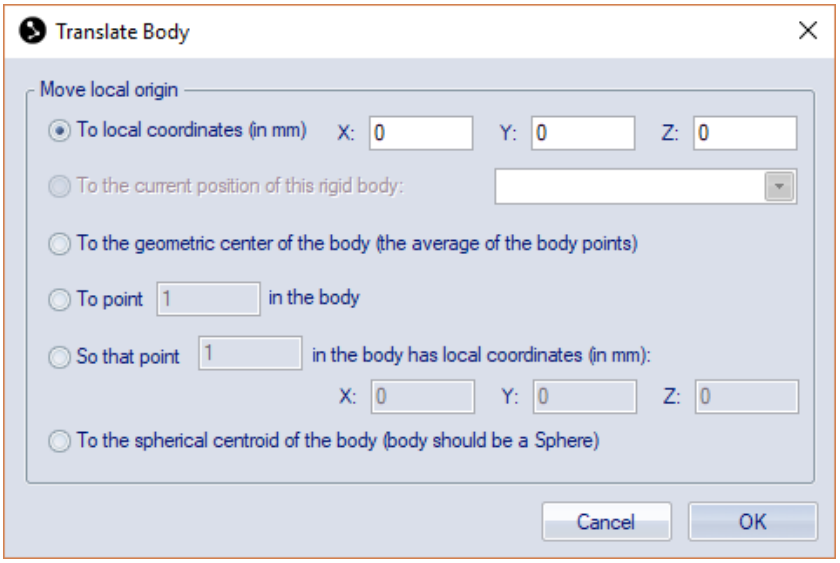
Load bodies
Loads bodies to the **Rigid bodies** list from an XML file.

Note: **Load bodies** will overwrite any other bodies in the list.

Save bodies
Save the bodies and all of the individual options in the **Rigid bodies** list to an XML file. Specify the name and the folder and click **Save**. The file can be edited in a text editor, e.g. Notepad. The xml format is the same as used for the RT protocol, see 6DOF xml parameters.

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. If you want to combine the rigid bodies from two or more different files you can copy-past them into a single file.

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.

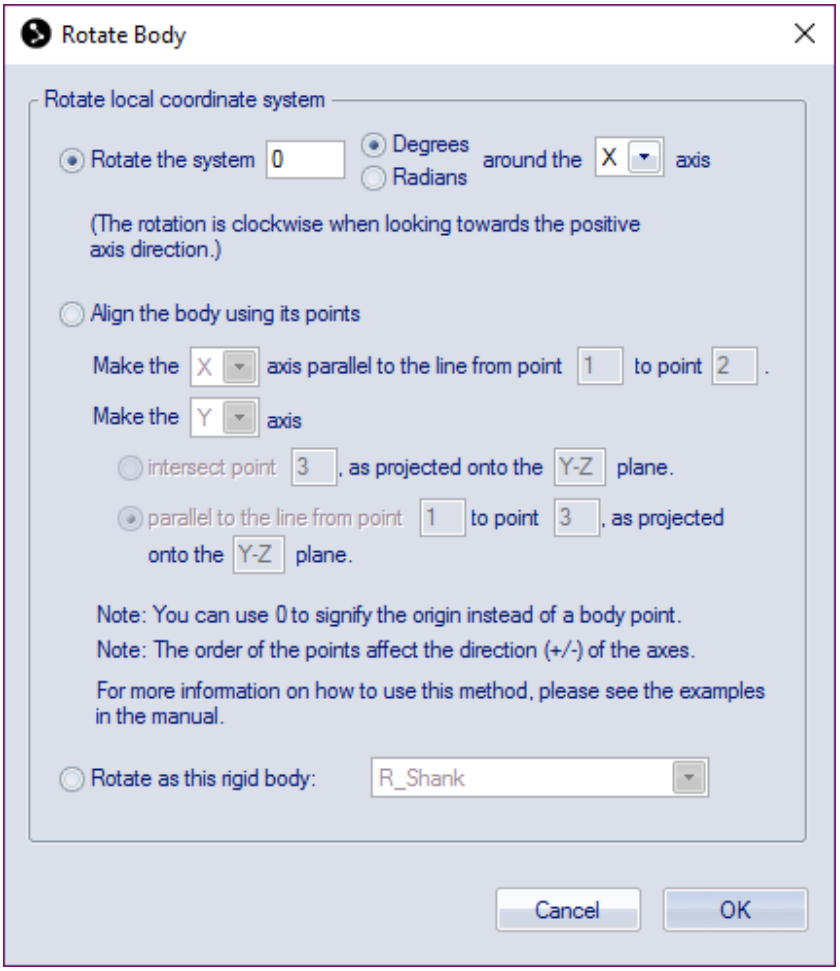
To the geometric center of the body (the average of the body points)
Move the local origin to the geometric center of all the points in the rigid body definition. The geometric center can be seen as the center of mass of a body with the same weight at all of the points.

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).

To the spherical centroid of the body (body should be a sphere)
Move the local origin to the spherical centroid of all the points in the rigid body definition. The points should be approximately configured in a sphere. The estimate of the centroid is based on a least square fit of the points.

Rotate body



The **Rotate body** dialog contains the following two ways to rotate the local coordinate system:

Rotate the system
Rotate the local coordinate system clockwise around one of the axes, when looking in the positive direction. Choose the angle of rotation either in **Degrees** or in **Radians**. Then select which axle to rotate round, **X**, **Y** and **Z**.
Note: The rotation are not changed by the Euler angles but will always be the same.

Align the body using its points
Define the rotation of the local coordinate system with three or four points from

the rigid body definition. You can use 0 to signify the origin of the local coordinate system if you want to use it as one of the points. Follow these steps to set the rotation:

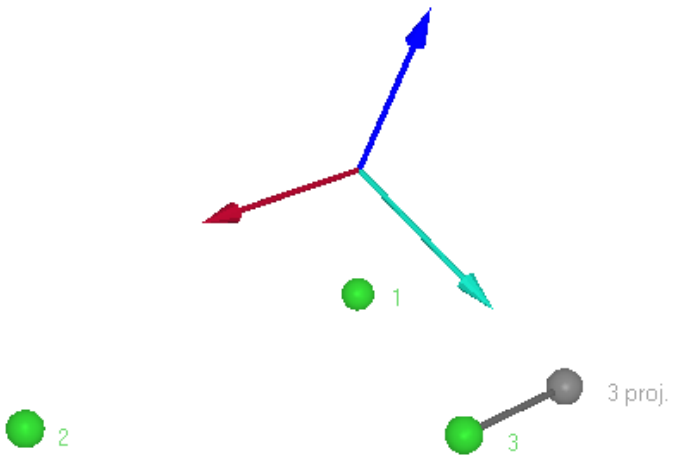
1. First start with defining one of the axes, choose which one from the drop-down list, by making it parallel to a line from one point to another, enter the number of the points in the body definition. The direction will depend on the order of the points so that the axis will always point in the direction from the first point to the other.

   Note: This does not mean that the origin of the body is placed in the first point.

2. Then you define the direction of a second axis, choose which one from the second drop-down list. Then there are two options for defining the second axes:

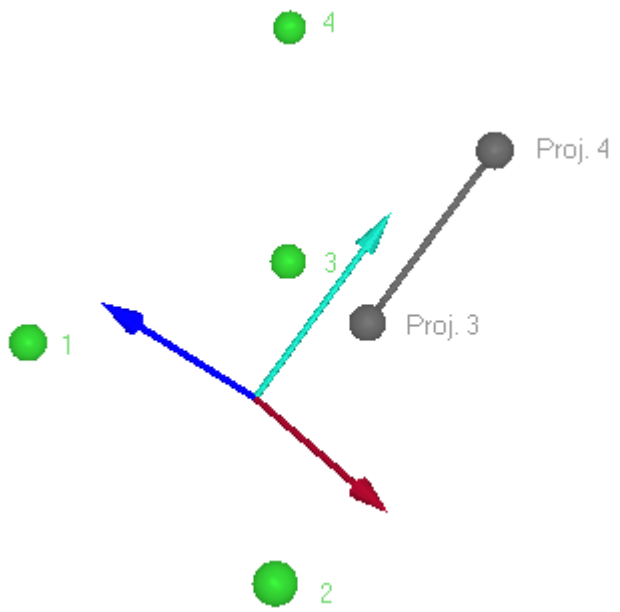
   Intersect point
   The intersect option means that the second axes will point in the direction of the specified point. However since the axes must be orthogonal, the second axes will actually intersect the projection of the point on the coordinate plane orthogonal to the first axes. The name of the orthogonal plane is displayed in the dialog.

   The example below displays the second axes defined by the projection of point 3 on the orthogonal plane defined by the line from point 1 to point 2.



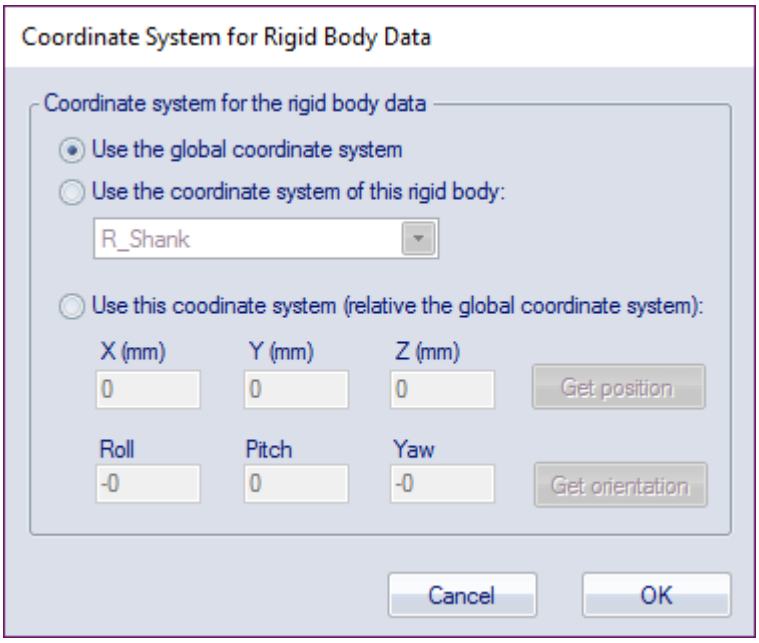
   Parallel to the line from point
   The parallel option means that the second axes will point in the direction from the first specified point to the second as projected on to the orthogonal plane to the first axes. The name of the orthogonal plane is displayed in the dialog.

   The example below displays the second axes defined by the projection of the line from point 3 to point 4 on the orthogonal plane defined by the line from point 1 to 2.



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**.

Smoothing 6DOF data

It is possible to smooth 6DOF data in QTM. The smoothing can be applied both in real time and in a capture. Smoothing of 6DOF data can be useful if you need to stabilize noisy data. The smoothing is applied to both position and orientation.

The smoothing algorithm includes two types of smoothing. The first type is "Holt-Winters" double exponential smoothing, taking into account the trend of the data. The second type is a jitter reduction filter, operating within a specified radius. In addition, there is a prediction parameter that can be used to project the trend ahead to compensate for lag. The parameters are:

- Data smoothing factor of the double exponential filter *s*, with a value between 0 and 1.
- Trend smoothing factor of the double exponential filter *c*, with a value between 0 and 1.
- Prediction parameter *p*, corresponding to the number of frames the trend is projected ahead.
- Position jitter radius *rp* in mm.
- Orientation jitter radius *ro* in degrees.

A disadvantage of smoothing is that it may introduce side effects. Smoothing and jitter reduction may introduce lag, whereas trend compensation may lead to overcompensation or overshoot. The smoothing effect of the filter, including the side effects may depend on the characteristics of the movement, as well as the capture rate. This should be taken into account when choosing the smoothing parameters.

QTM includes presets with useful combinations of smoothing parameters for various situations. The presets can be selected individually per rigid body. The available filter presets are:

No filter
No smoothing. This is the default option.

Multi-purpose
Light smoothing and jitter reduction. This preset is suitable for most situations which require effective smoothing of light noise, while keeping side effects to a minimum. The filter parameters are: *s*=0.25, *c*=0.25, *p*=0, *rp*=5, *ro*=5.

High stability
Strong smoothing and jitter reduction. This preset is suitable when you need stable data in challenging tracking situations. In case of high accelerations or impacts filtering may lead to noticeable side effects. The filter parameters are: *s*=0.5, *c*=0.5, *p*=1, *rp*=25, *ro*=25.

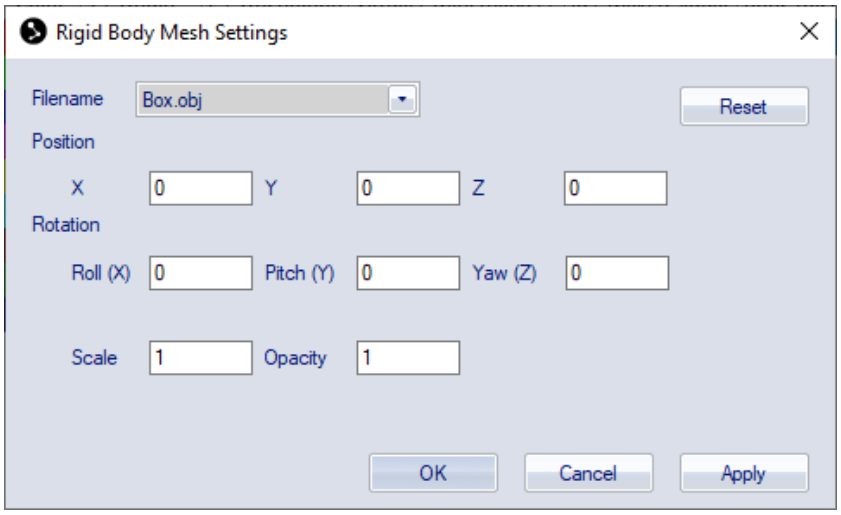
Static pose
Effective jitter reduction for immobile objects. Application of this preset to moving rigid bodies may lead to noticeable lag for small displacements and

rotations within the specified radius. The filter parameters are: *s*=0, *c*=0, *p*=0, *rp*=10, *ro*=10.

The selected presets in the rigid body definitions in the **Project Options** will be applied in real time and in subsequent captures when **Calculate 6DOF** is checked in the **Processing** options. It is also possible to reprocess captures with different presets.

The effect of the smoothing filter can be visually inspected in the **3D view** window by comparing the markers of the rigid body overlay in the 3D view window with the measured marker positions. For more detailed comparison of the 6DOF data, you can define the same rigid body twice, one with filter and the other one without.

Rigid body Mesh Settings dialog



The **Rigid Body Mesh Settings** dialog is used to associate a mesh with a rigid body and to inspect and modify the mesh object settings. For more information about how to use Rigid body meshes see chapter “Rigid body meshes” on page 347.

The dialog contains the following settings:

Filename
A drop-down list of mesh objects in the *Meshes* folder of the project. Copy the .obj file, and any associated .mtl or image file, manually to the *Meshes* folder to use the mesh in the project. The path to the *Meshes* folder can be set on the Folder options page, see chapter “Folder options” on page 250. For information about obj files and the features supported see chapter “Compatibility of meshes” on the next page.

Position
The 3D translation of the object relative to the global coordinate system. The values of **X**, **Y** and **Z** are specified in mm units.

Rotation
The 3D rotation of the object relative to the global coordinate system. The values for **Roll**, **Pitch** and **Yaw** are specified in degrees around the X, Y and Z axes, respectively.

Scale
The scale factor of the mesh object.

Opacity
The opacity of the mesh object from 0 (transparent) to 1 (opaque). An .obj file can include an opacity option for individual faces, these will be rendered as transparent even if the **Opacity** option is set to 1.

The buttons:

Reset
Remove the link to the file and resets the **Position**, **Rotation** and **Scale** values.

OK
Accept the settings and close the dialog.

Cancel
Discard the settings and close the dialog.

Apply
Review the changes directly in the **3D View** window while in **Preview** mode.

Compatibility of meshes

A mesh file consists of a lot of faces and each individual face can be associated with materials and textures, but not all of the features applied to a face is rendered in QTM. Complex obj files may not be supported and then those faces and textures are not rendered. Below are some of the features that are not supported.

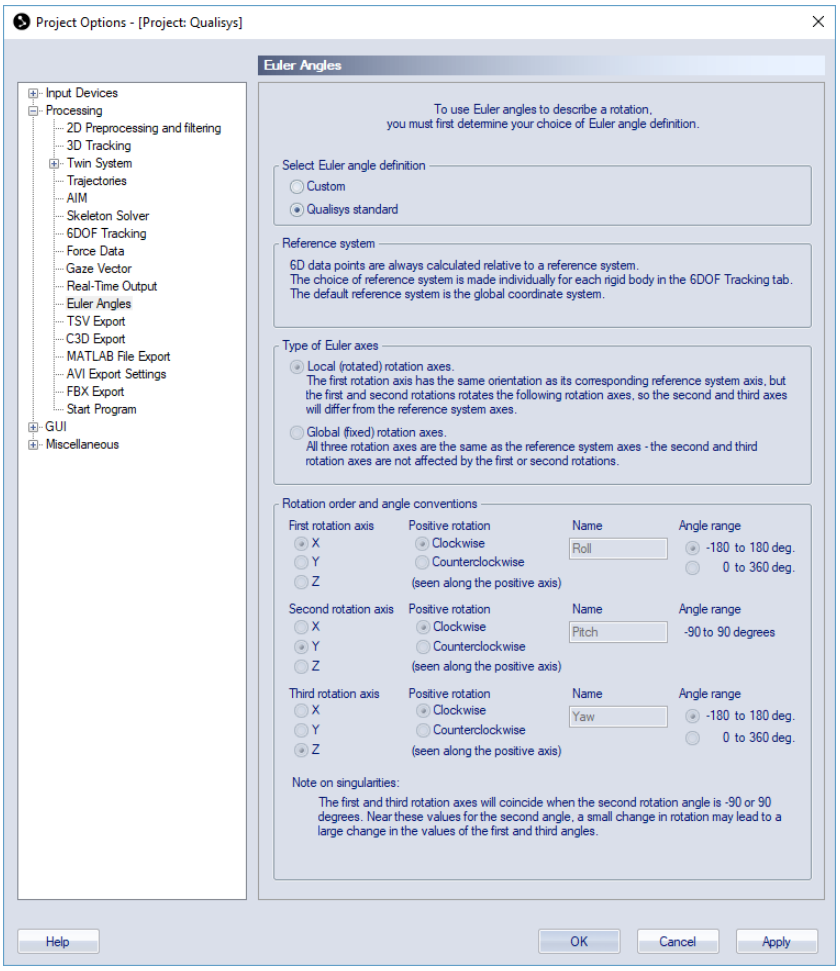
1. The material is only rendered for the front side of a face. If a material doesn't show the reason can be that it is applied to the backside of the face.
2. Only four faces can be included on one row in the .obj file. If there are more than four faces on a row then none of them are rendered.
3. The paths to the material and texture files are sometimes written as absolute paths in the .obj and .mtl file. So if the material or texture is not applied to the mesh, check that the paths are relative and the file names are correct.

Euler angles

On the **Euler angles** page you can change the definition of the Euler angles used in QTM. The definition applies to all places in QTM where Euler angles are used, displayed or exported, including:

- Rotation of the global coordinate system
- The orientation rigid bodies (6DOF)
- The orientation of skeleton segments

Select Euler angle definition

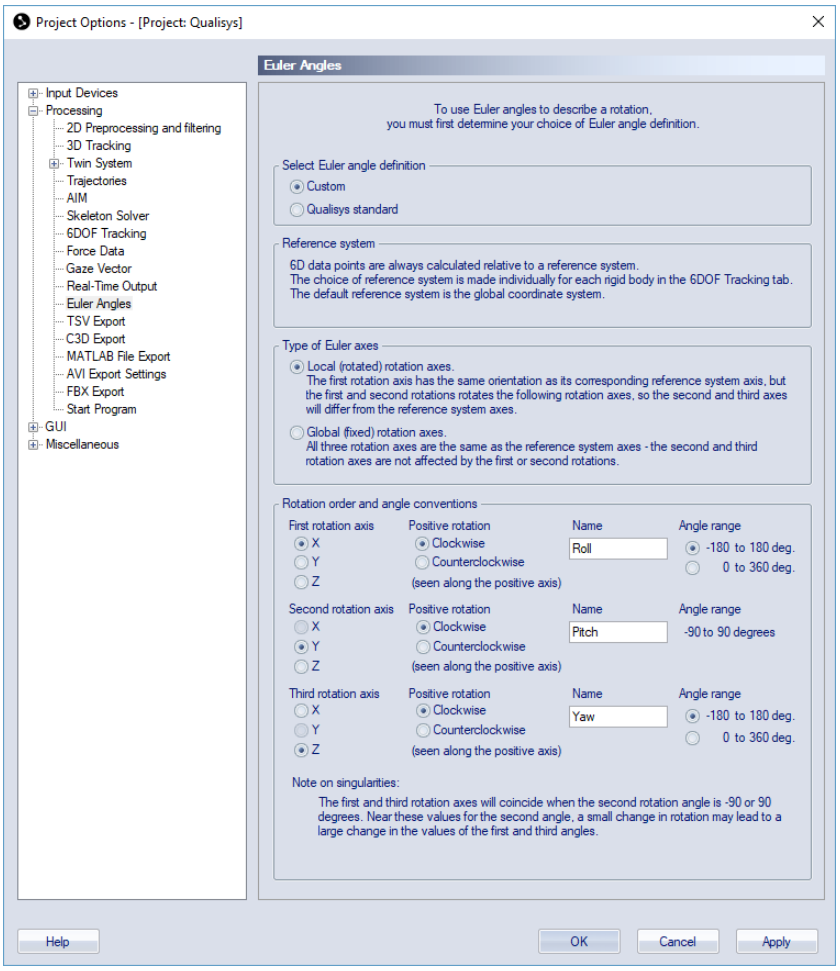


By default QTM uses the **Qualisys standard**, which is described in the chapter "Rotation angles in QTM" on page 344. The definition can also be seen when **Qualisys standard** is selected as the grayed settings under the **Definition of rotation axes** heading, see screen dump above.

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°.

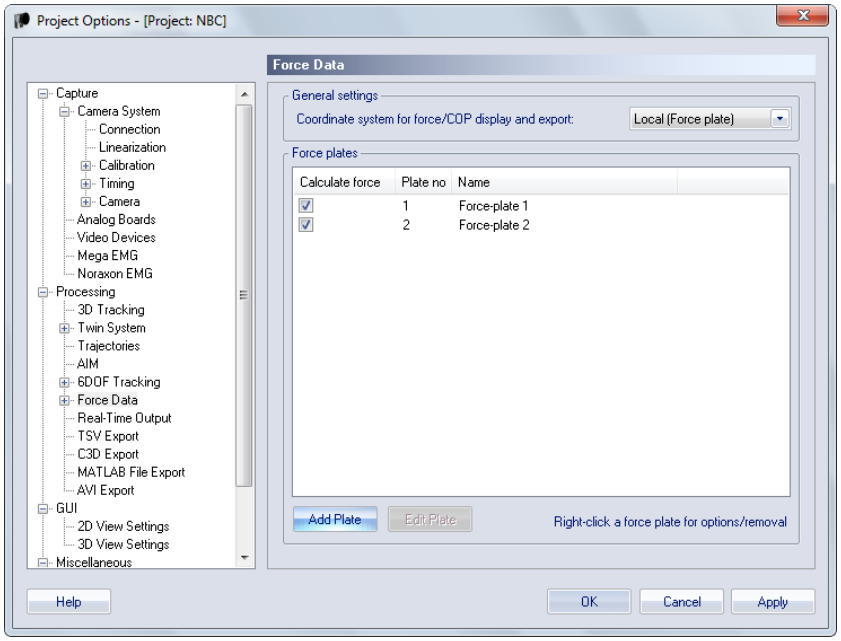
   First and third rotation round the same axis

   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 386.



General settings

The setting **Coordinate system for force/COP display and export** defines in what coordinate system the force data is displayed in the **Data info** window and exported.

Note: The C3D export only includes analog data and force plate parameters, so it is therefore not changed by this setting.

The default value is **Local (Force plate)**, which means the local coordinate system of the force plate with the origin in the middle of the force plate.

You can change the orientation to **World (Lab)** and then all of the force data will be in the global coordinate system of the measurement.

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. This applies both to force plates connected to an analog board, Kistler DAQ and AMTI Gen5.

Use the **Add plate** option to add a new force plate to the list. Right-click on the force plate to open a menu where you can **Change name** and **Remove plate**. The AMTI Gen5 plates are created automatically and cannot be removed. They can however be renamed and then the same name is used for it on the **Input Devices** page.

To enable force plates in QTM, select the check box next to the force plate name in the **Calculate force** column.

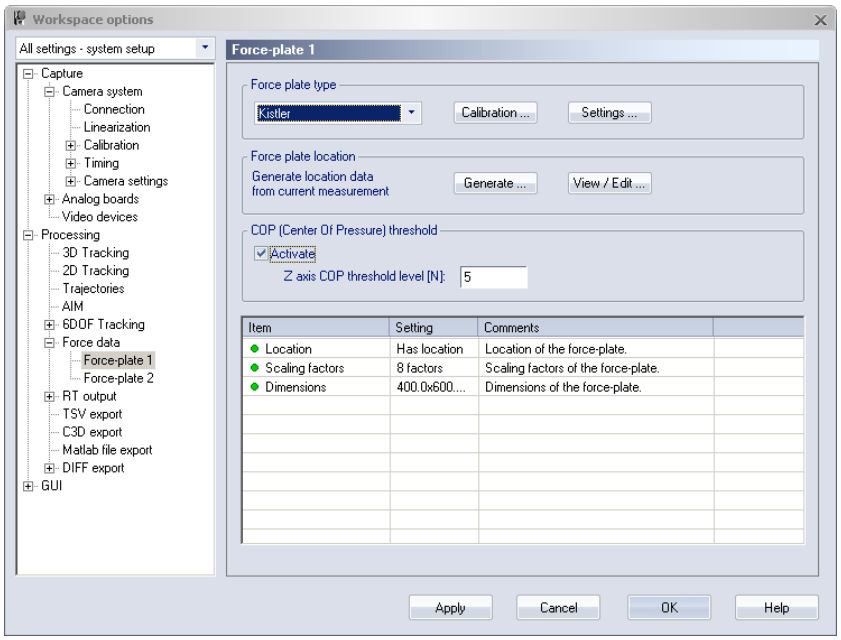
Select a force plate and click **Edit plate** to open the settings for that plate. The settings is also opened by double-clicking on its row. The settings differs between the different force plate types, see chapter “Force plate” below. For more information about force plates see chapter “How to use force plates” on page 454.

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 connected to an analog board or Kistler DAQ, the devicethat it is connected to must be activated 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 six different types available for force plates connected to an analog board: **AMTI**, **Kistler**, **Bertec**, **Portable AMTI** , **QMH**, **Treadmetrix**, **Generic 6 ch (c3d type-2)**, **Generic 8 ch (c3d type-3)** and **Generic 6 ch with matrix (c3d type-4)**.

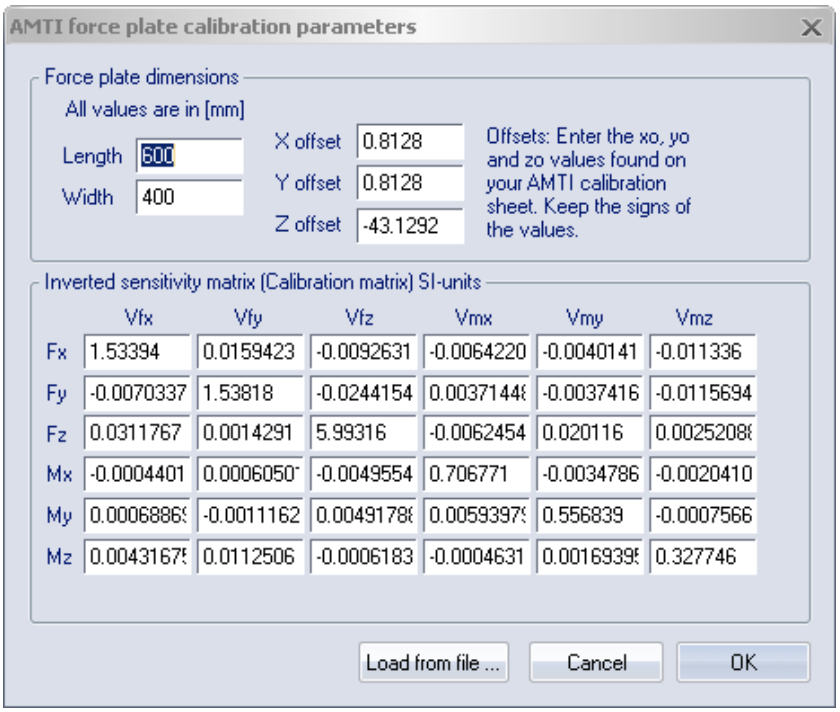
For the AMTI Gen5 force plates there is only the **AMTI Gen5** type. The Calibration and Settings options are controlled automatically, if the force plate has a new type of calibration. If the force plate uses the old calibration matrix then you need to edit the matrix in the AMTI Netforce program before you can use the plate in QTM.

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.

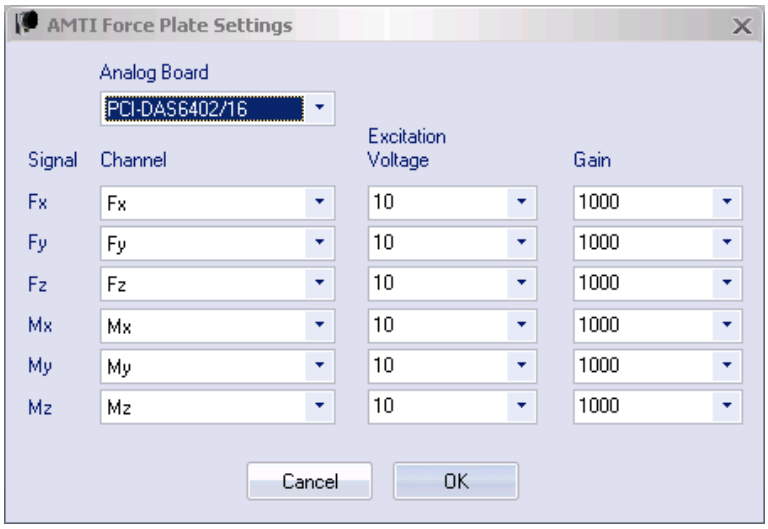
Inverted sensitivity matrix

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.

Note: The file contains the Sensitivity matrix, which is then converted to the inverted when imported to QTM.

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.

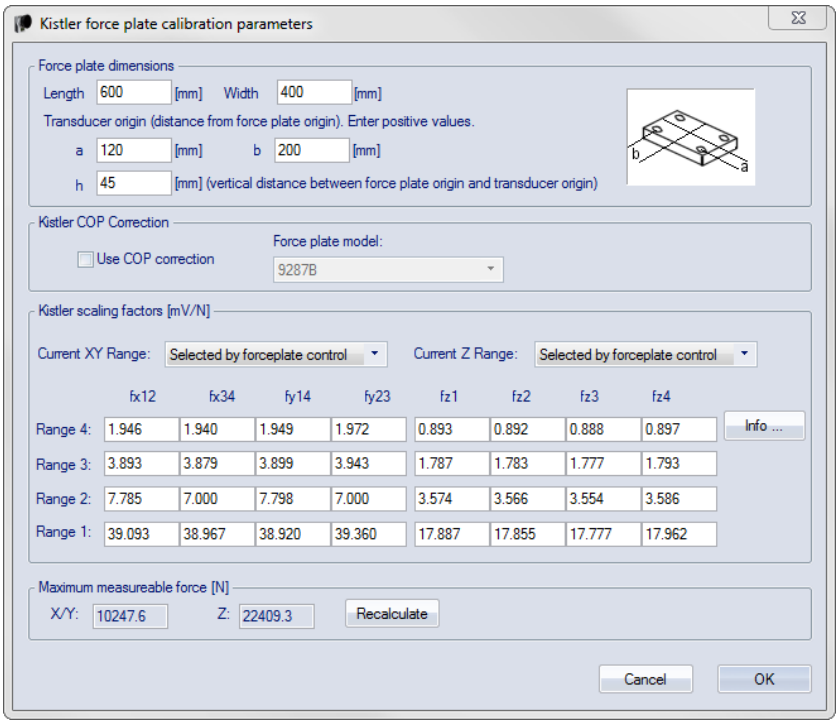
Note: The drop-down list gives you standard values, but you can also type any value for the setting.

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 length, width and transducer origin parameters for the Kistler force plate. These parameters can be found in the documentation that is delivered with the Kistler force plate. If you cannot find these values, please refer to Kistler support.

Note: Enter the absolute values of the parameters, that is for the h value write 45 when the Kistler manual says -45.

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. The scaling factor ensures that the force data is scaled correctly. Follow these steps to use the correct scaling factors.

Note: If you update from a version earlier than QTM 2.7 all of the ranges will get the value of the currently entered range. Enter the correct values for all of them so that you can switch ranges more easily.

1. How to find the scaling factors differs between internal and external amplifiers.

   Internal amplifier
   For a Kistler force plate with internal amplifier enter all of the scaling factors that are found in the calibration certificate matrix of each force plate.

   External amplifier
   For a Kistler force plate with external amplifier you must calculate the scaling factors, see instructions below.

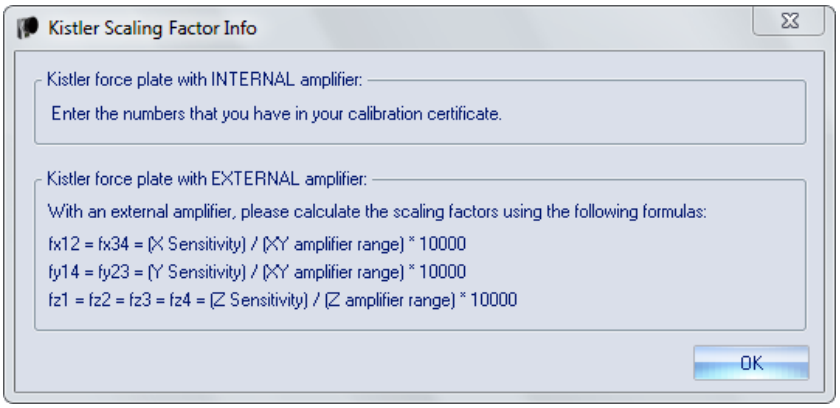
2. Then select the range with the **Current XY range** and **Current Z range** options:

   Selected by forceplate control
   The default option is **Selected by forceplate control**, which means that the range is controlled by the ranges set on the **Force plate control settings** page.
   Note: The force plates will be ordered by the analog channels that they are connected to, so that the one with the first channels will use the ranges of the first plate on the **Force plate control settings** page.

   Range 4 - Range 1
   You can select the range manually if you don't use the analog board to control the force plate ranges. Range 4 will give you the maximum force range and range 1 will give you highest sensitivity, i.e. minimum force range. It is important to use the same range as is used by the force plate to get correct forces, you can use different ranges for the XY and Z settings.

Calculating scaling factors with external amplifier

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, click on **Info** for more information. Calculate the scaling factors for all of the ranges with the help of the formulas in the **Kistler scaling factor info** dialog. The force plate sensitivity parameters and the value of the ranges are found in the Kistler calibration sheet.

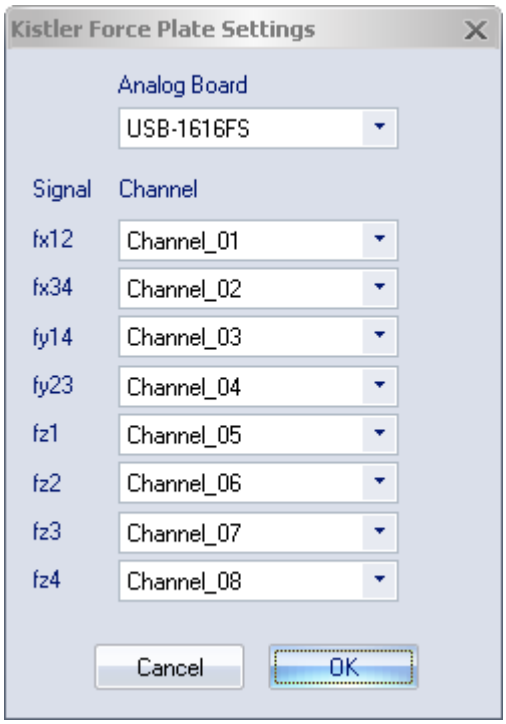


Maximum measurable force

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.

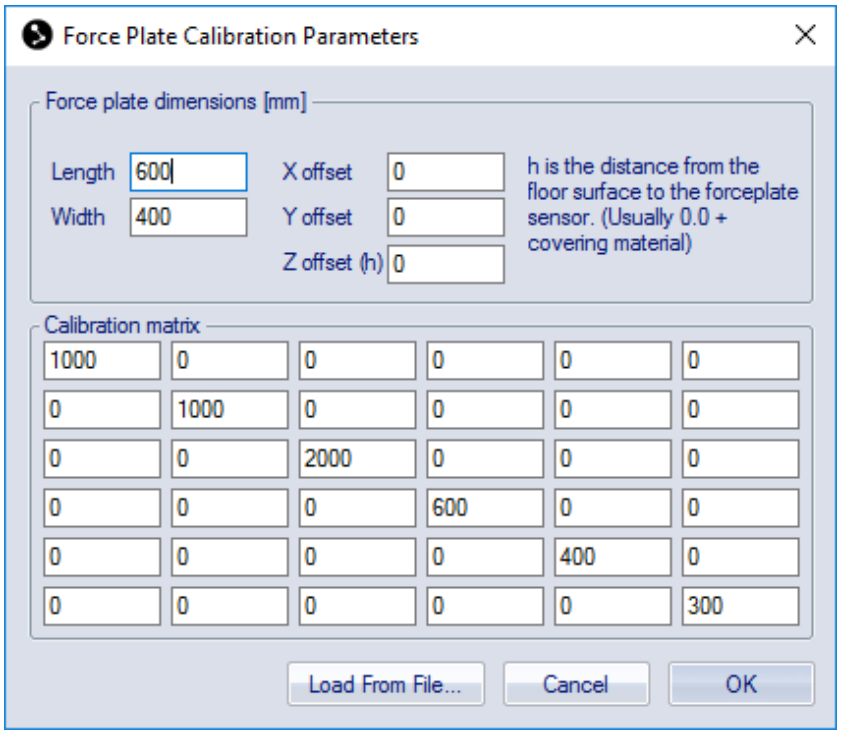
Then associate each signal from the force plate with its respective analog channel with the **Channel** settings.

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

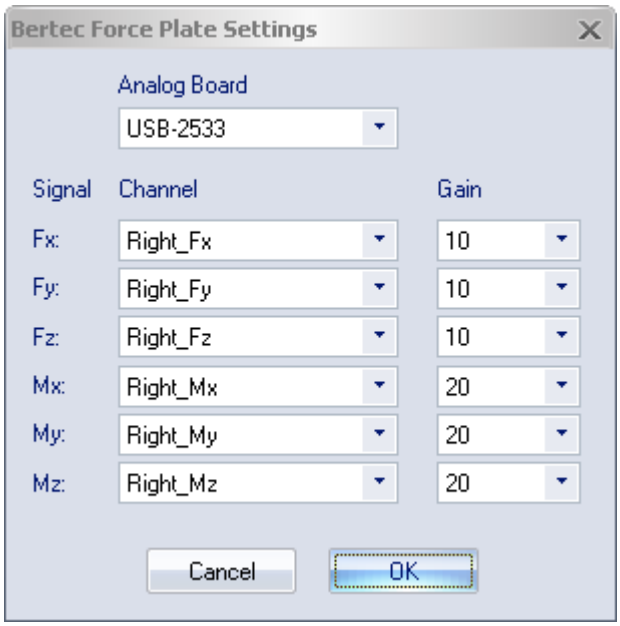
Under the **Force plate dimensions** heading you should enter the dimensions parameters for the Bertec force plate. The parameters can be found in the user manual of the Bertec force plate.

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.

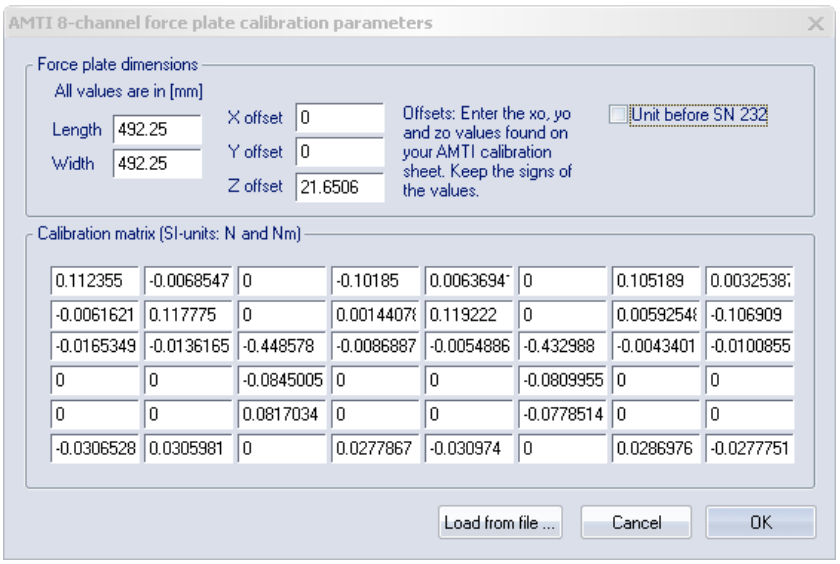
Note: On the **Analog board (...)** page the channel names can be renamed to match the signal names.

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.

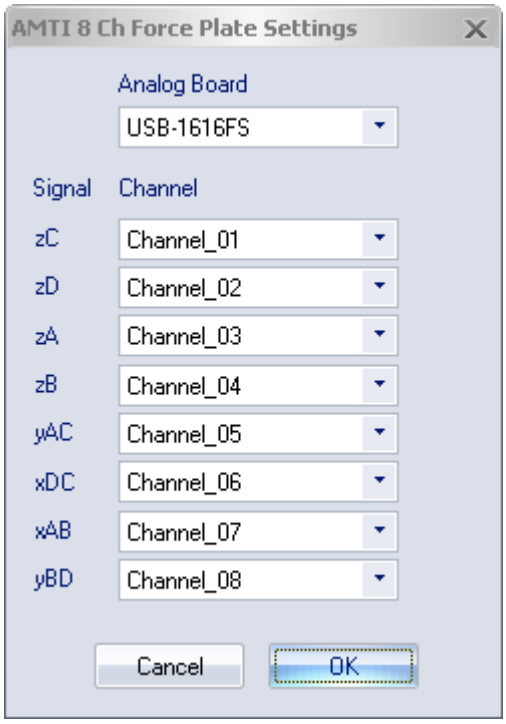
Old type of force plate
Activate the **Unit before SN 232** option, if you have a force plate with serial number lower than 232. These force plates have a "zero" level of 2.5 V. When calculating the force in a file, the data is always zeroed with the first 10 frames of the measurement. This is because 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.
Note: The file contains the Sensitivity matrix, which is then converted to the inverted when imported to QTM.

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.

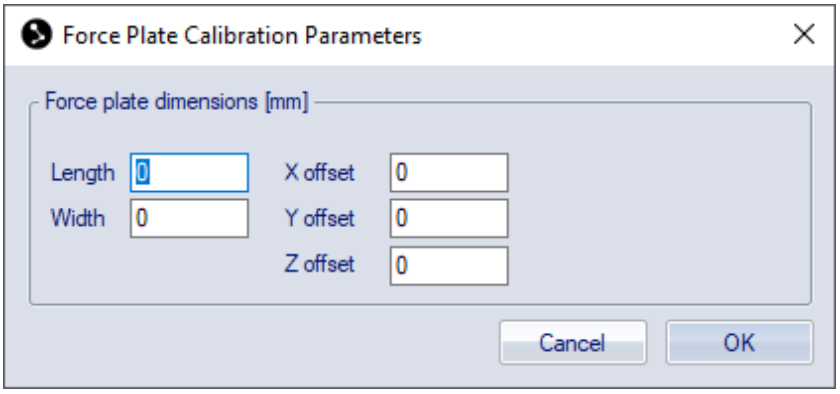
For more information about the signals see the manual of the AMTI portable force plate.

Generic 6 ch (c3d type-2) force plate calibration parameters

Select the Generic 6 ch (c3d type-2) force plate type and click **Calibration** under the **Force plate type** heading to go to the **Force plate calibration parameters** dialog. It contains the settings for dimensions of the Generic 6 ch (c3d type-2) force plate.

Note: The main use of the Generic 6 ch (c3d type-2) force plate is for handling of imported c3d files. It is recommended to use the vendor specific force plate types when capturing data.

Note: For information about c3d type-2 plates refer to the c3d.org website.



Force plate dimensions

Under the **Force plate dimensions** heading you should enter the dimension parameters (in mm) for the Generic 6 ch (c3d type-2) force plate. The parameters can be found in the user manual of the force plate.

Generic 6 ch (c3d type-2) force plate settings

Select the Generic 6 ch (c3d type-2) force plate type and click **Settings** under the **Force plate type** heading on the **Force plate** page to open the **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.

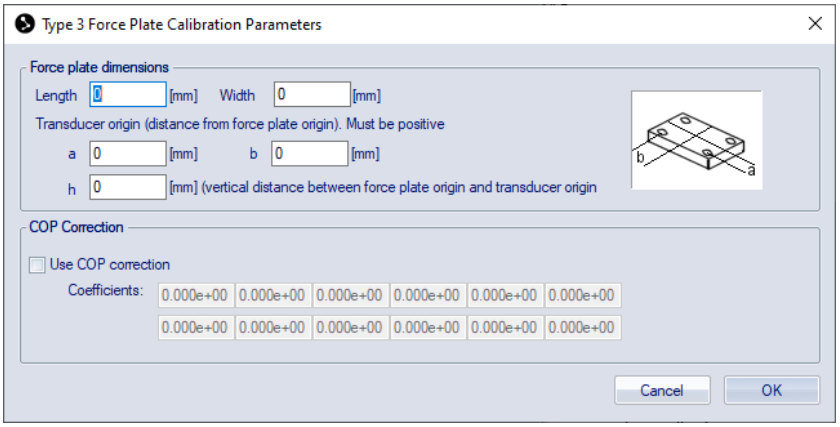
For more information about the signals see the manual of the force plate.

Generic 8 ch (c3d type-3) force plate calibration parameters

Select the Generic 8 ch (c3d type-3) force plate type and click **Calibration** under the **Force plate type** heading to go to the **Type 3 Force plate calibration parameters** dialog. It contains the settings for dimensions of the Generic 8 ch (c3d type-3) force plate.

Note: The main use of the Generic 8 ch (c3d type-3) force plate is for handling of imported c3d files. It is recommended to use the vendor specific force plate types when capturing data.

Note: For information about c3d type-3 plates refer to the c3d.org website.



Force plate dimensions

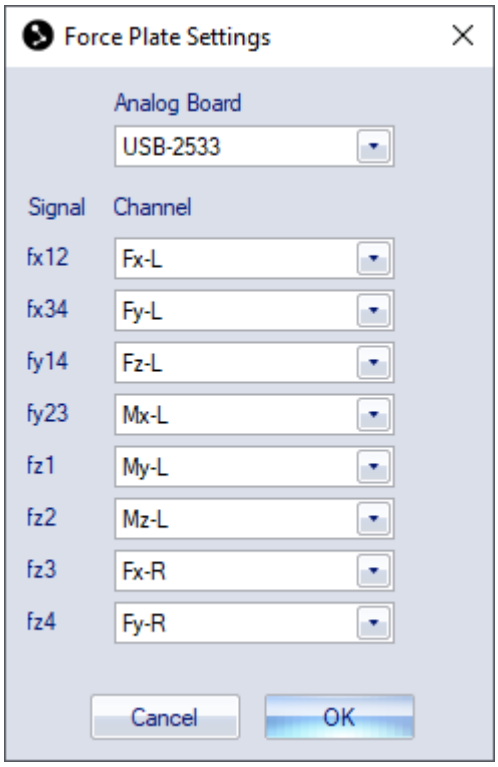
Under the **Force plate dimensions** heading you should enter the dimension parameters (in mm) for the Generic 6 ch with matrix (c3d type-4) force plate. The parameters can be found in the user manual of the force plate.

COP correction

Under the **COP correction** heading you can enable COP correction with the **Use COP correction** option. Enter the **Coefficients** for the correction polynomial.

Generic 8 ch (c3d type-3) force plate settings

Select the Generic 8 ch (c3d type-3) force plate type and click **Settings** under the **Force plate type** heading on the **Force plate** page to open the **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.

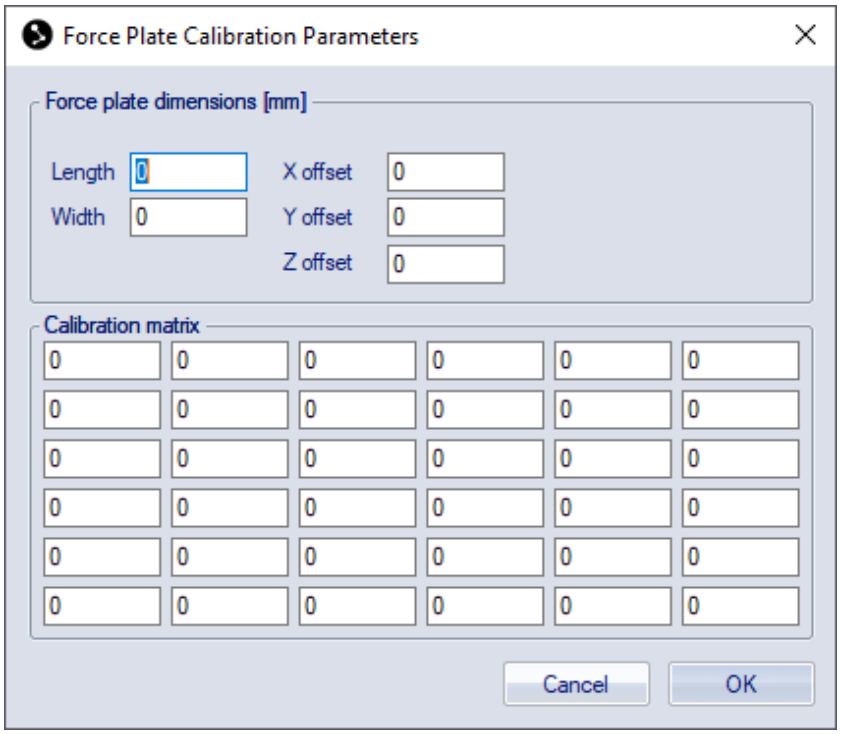
For more information about the signals see the manual of the force plate.

Generic 6 ch with matrix (c3d type-4) force plate calibration

Select the Generic 6 ch with matrix (c3d type-3) force plate type and click **Calibration** under the **Force plate type** heading to go to the **Force plate calibration parameters** dialog. It contains the settings for dimensions of the Generic 6 ch with matrix (c3d type-3) force plate.

Note: The main use of the Generic 6 ch with matrix (c3d type-3) force plate is for handling of imported c3d files. It is recommended to use the vendor specific force plate types when capturing data.

Note: For information about c3d type-4 plates refer to the c3d.org website.



Force plate dimensions

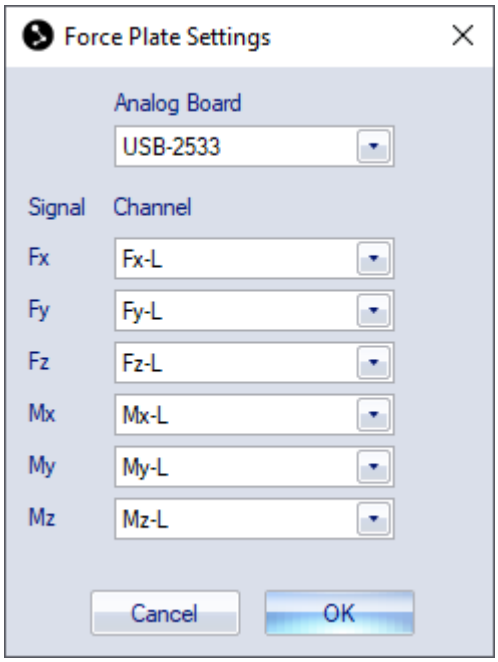
Under the **Force plate dimensions** heading you should enter the dimension parameters (in mm) for the Generic 6 ch with matrix (c3d type-4) force plate. The parameters can be found in the user manual of the force plate.

Calibration matrix

The **Calibration matrix** is used to calibrate the force plate. Enter the values of the calibration matrix (**M**) (in SI-units) under the **Calibration matrix** heading. The values can be found in the manual of the force plate. The calibration matrix must fit in this formula **F** = **M** * **A**

Generic 6 ch with matrix (c3d type-4) force plate settings

Select the Generic 6 ch with matrix (c3d type-4) force plate type and click **Settings** under the **Force plate type** heading on the **Force plate** page to open the **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.

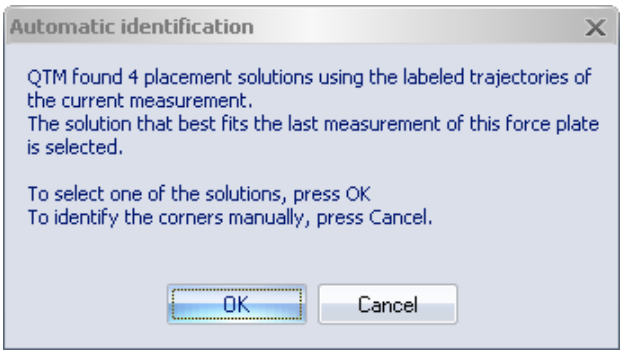
For more information about the signals see the manual of the force plate.

Force plate location

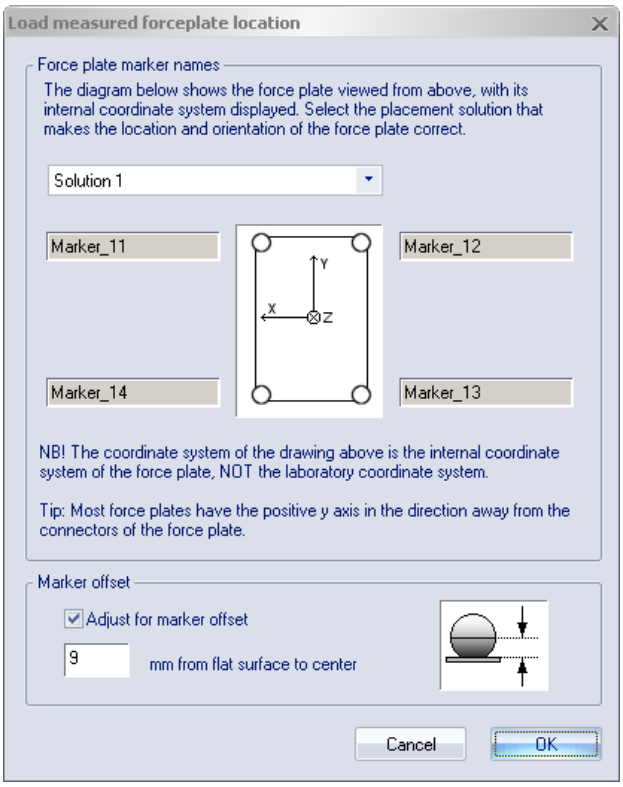
To be able to view the force vectors in the same coordinate system as the motion capture, the location of the force plate must be specified. The settings are the same for all force plate types and are done under the **Force plate location** heading on the **Force plate** page. The force plate location 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.
   a. Make a 1 second motion capture of these markers.
   b. Identify the markers, you can give them any label, and keep the capture file open.
   c. 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:



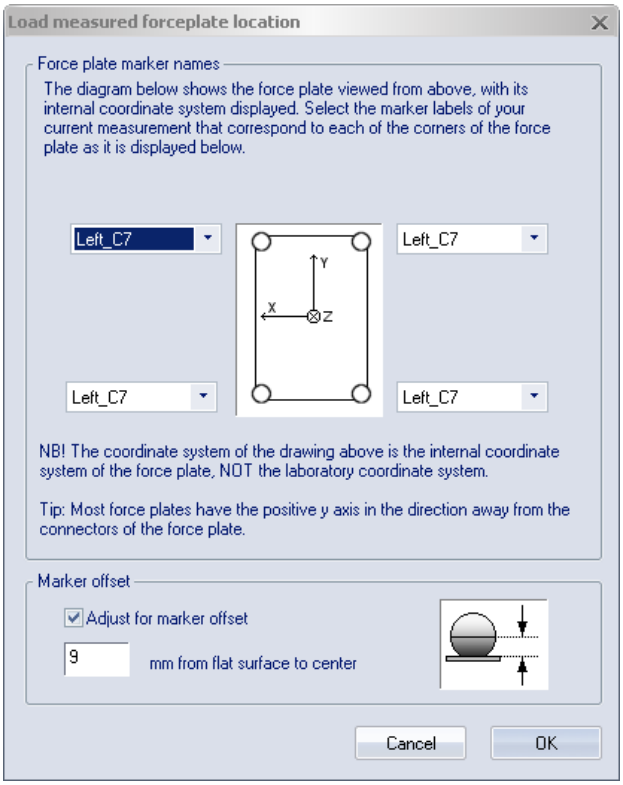
d. Click **OK** to open **Load measured force plate location** dialog, see below, and select one of the solutions found by QTM. Click **Cancel** to select the markers' locations manually, see further down. Try to make sure that the orientation is correct. It is recommended to make a test measurement 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.



e. 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:

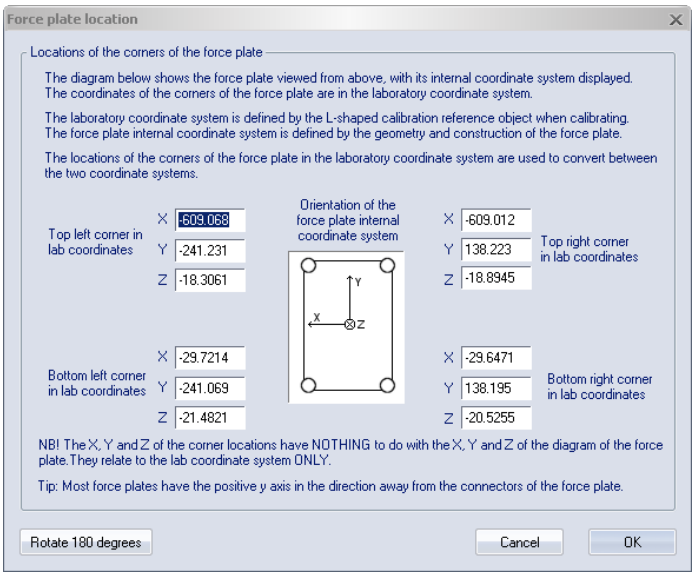
Marker size	Adjustment
12 mm	5 mm
19 mm	8 mm
30 mm	13 mm

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. When the Z component of the force is below the threshold level, the force vector will not be shown in QTM. The COP will be calculated though and accessible in the file.

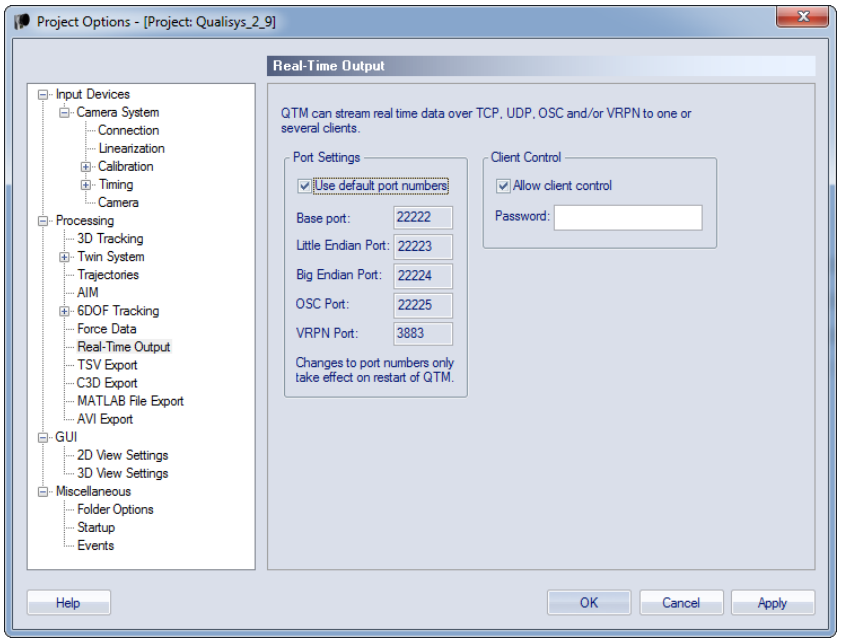
Select the **Activate** check box to activate the COP threshold filter. Enter the **Z axis COP threshold level** in Newton to disable the visualization of the force vector and COP. The Z axis is in the force plate coordinate system, which means that it is always the vertical force which is used in the 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**.

Real-Time 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.



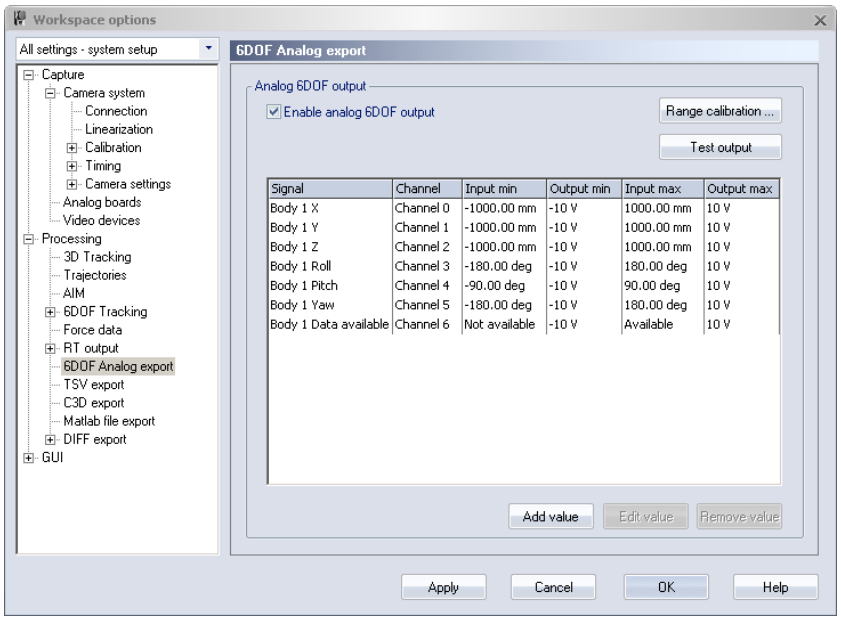
The setting that can be changed for the RT output is the TCP/IP and OSC port numbers. Uncheck the option **Use default port numbers** to set other port numbers. The first four ports are grouped together, so that their port number are changed with the **Base Port** number, which by default is 22222. The **VRPN Port** number can be changed independently of the others and is by default 3883. For a description of the different ports see the protocol 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.

6DOF analog export

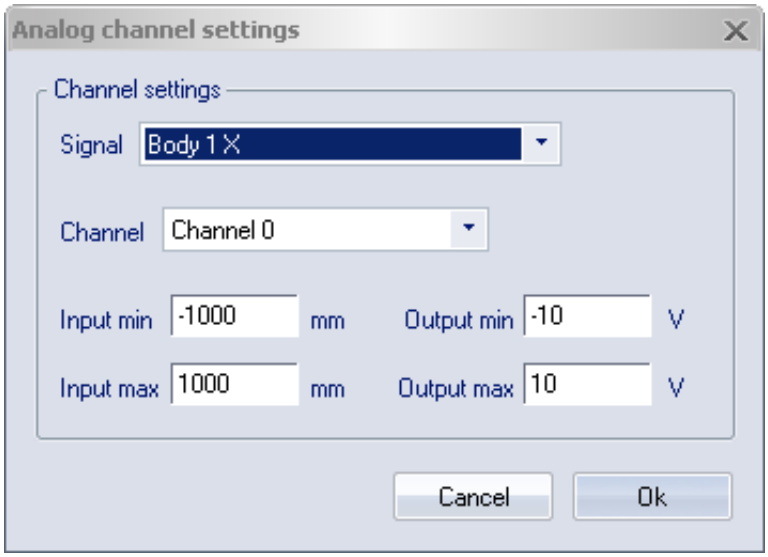
The **6DOF analog export** page is only available if an analog output board (PCI-DAC6703) is installed in the measurement computer. With the analog export the information about 6DOF bodies' positions can be used in feedback to an analog control system. Select the **Enable analog 6DOF output** option 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.

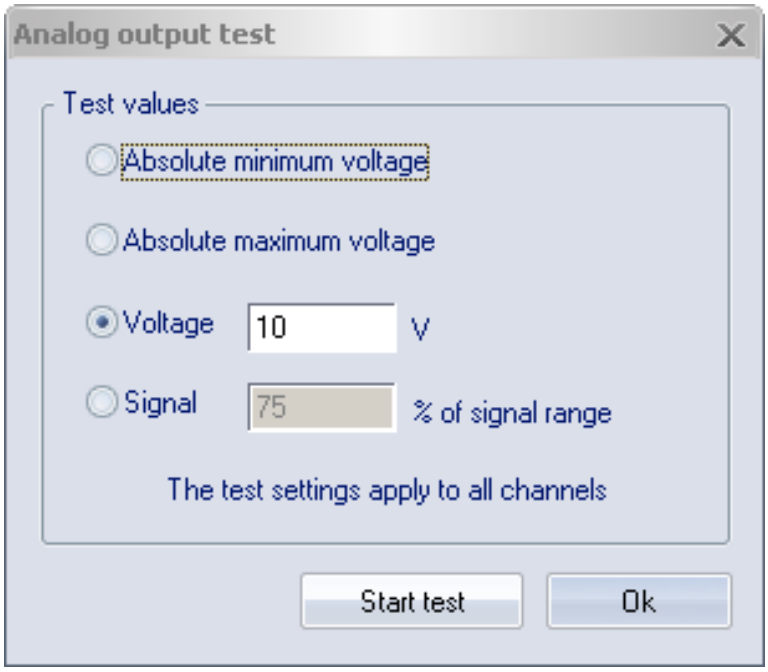
Input min/Input max
The minimum respectively the maximum value (mm or degrees) of the input data. If the input data is equal or smaller than the **Input min** the output of the channel is equal to the **Output min**. If the input data is equal or larger than the **Input max** the output of the channel 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.

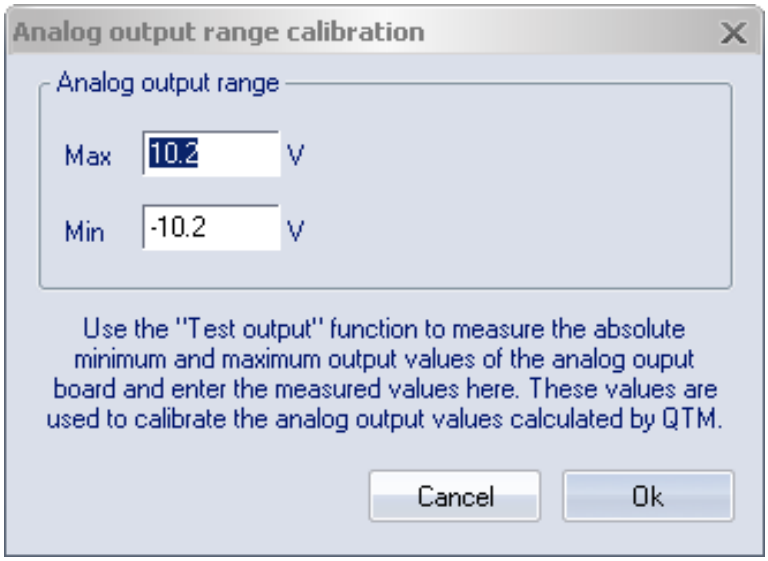
Absolute maximum voltage
The output of all channels are set to the maximum value of the analog board. This value should be entered on the **Analog output range calibration** dialog.

Voltage
The output of all channels are set to the specified voltage.

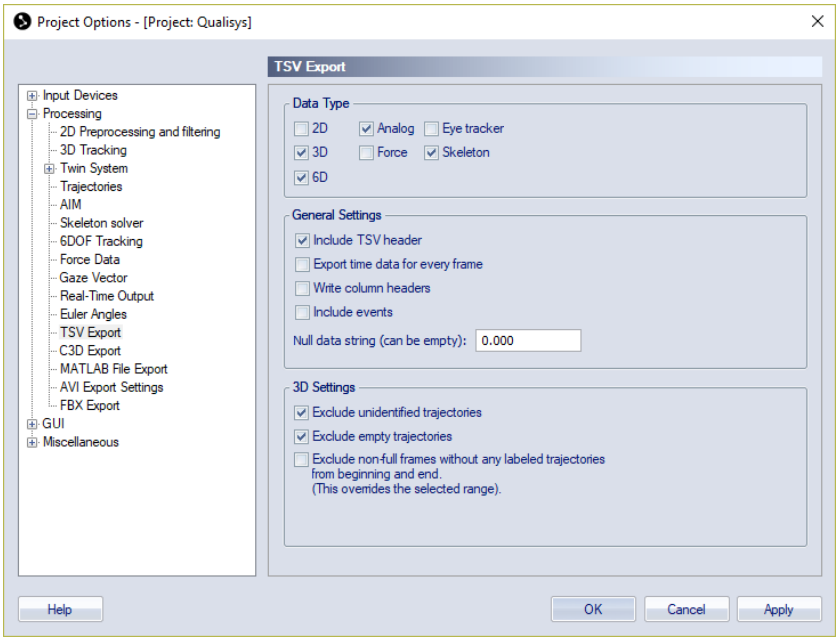
Signal % of signal range
The output of all channels is set to the specified percentage of the channel's specified output range. If the channel is not used the output will be 0 V.

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 389.

Data type to export

Under the **Data type to export** headings you can select which data types to export by checking the items. Each data type is exported as a separate file with a suffix added to

the file name. The following types of motion data can be selected:

2D data
Export the 2D data of the capture file.

3D data
Export the 3D data of the capture file.

6D data
Export the 6DOF data of the capture file.

Analog data
Export the analog data of the capture file.

Force data
Export the force and moment data of the capture file.

Eye tracker data
Export eye tracker data of the capture file.

Skeleton data
Export skeleton data of the capture file.

For more detailed information about the respective file formats, see chapter “TSV file formats” on page 390.

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 or 3D TSV file.

Settings for 3D data export

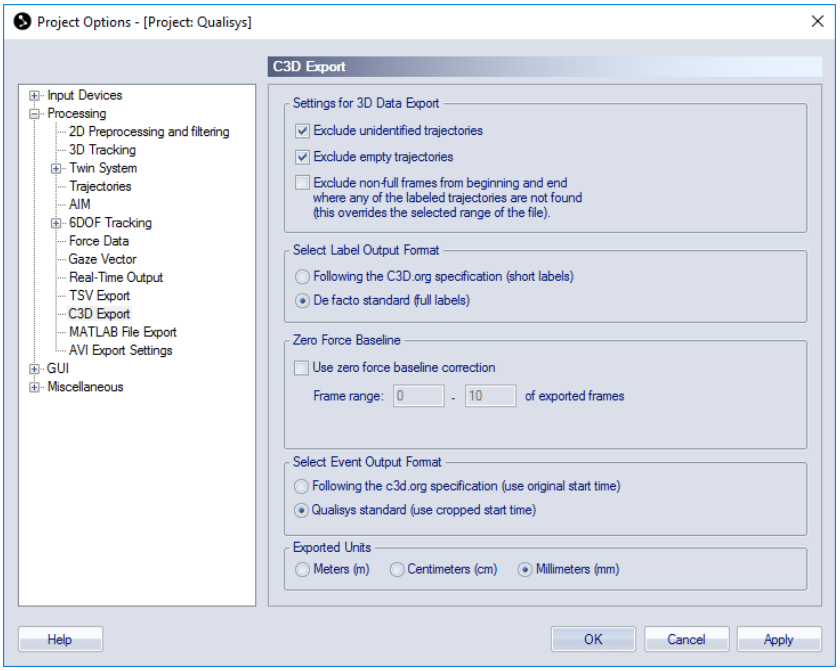
The settings under the **Settings for 3D data export** heading are only applied to 3D data. The settings are:

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 397.

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.

Select event output format

Under the **Select event output format** heading you can change the format of the C3D file with the following two settings:

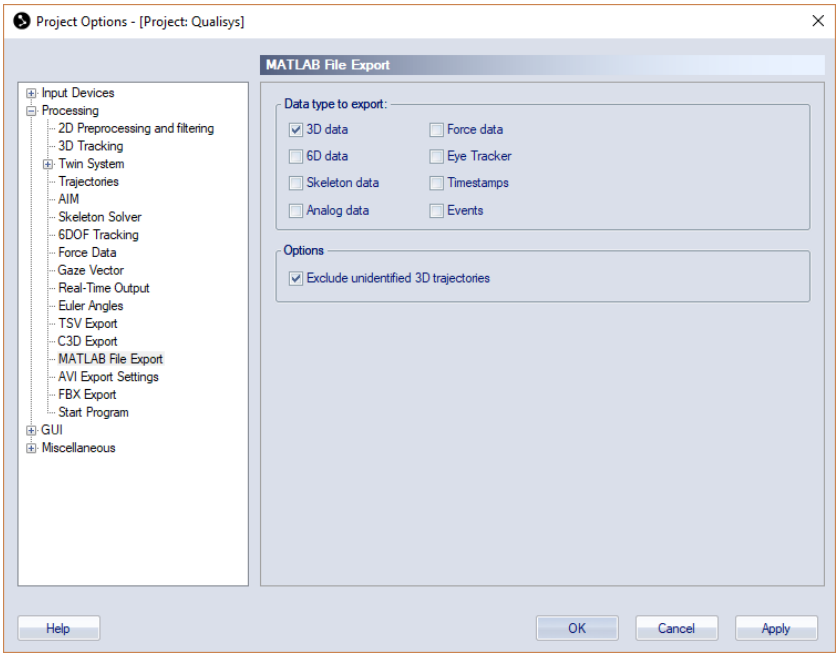
Following the C3D.org specification (use original start time)
Use the C3D.org specification, which uses original start time.

Qualisys standard (use cropped start time)
Event time specified relative to cropped start time.

Exported units

Specify the length unit used for the C3D export. The alternatives are meters (m), centimeters (cm) and millimeters (mm, default). This setting applies to all length data, including 3D trajectories and force plate positions.

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 399. Select which data types 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. Use the **Exclude unidentified trajectories** option to only export the labeled trajectories to the MAT file.

6D data
Include 6DOF data.

Skeleton data
Include skeleton data.

Analog data
All of the analog data will be included, both data from analog boards and EMG.

Force data
Include force data.

Eye Tracker
Include eye tracker data.

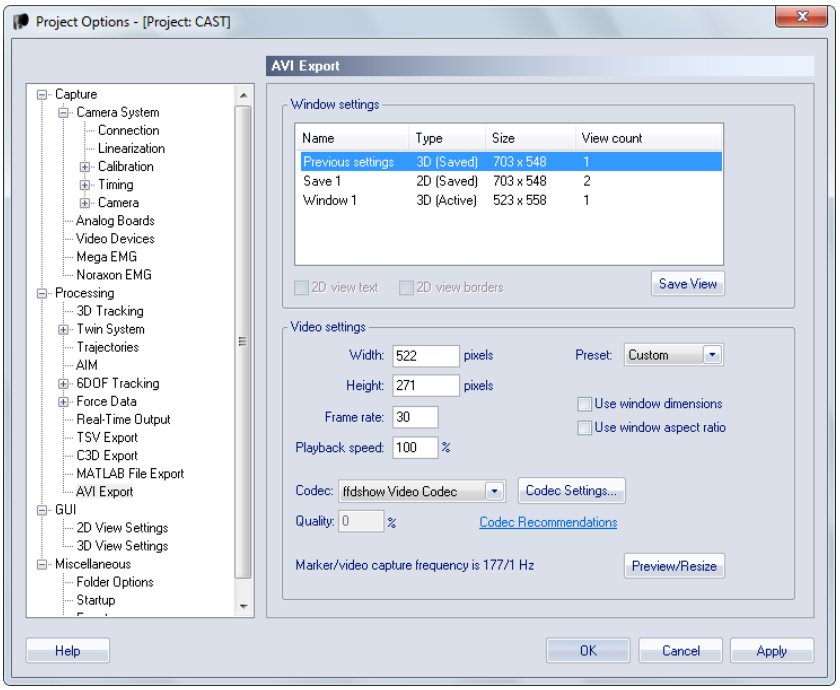
Timecodes
Include timestamps for each camera frame, according to **Timestamp** settings.

Events
Include events.

AVI Export

The **AVI Export** page contains settings for the export of 3D or 2D views to AVI files. The export can be done either as a processing step or manually from the **File/Export** menu. For more information see chapter “Export to AVI file” on page 404.

Note: If you export an AVI file manually the current setting of the processing step will also change. Therefore it is recommended that you save the View that you want to use in a processing step so that you can return to that setting.



Use the settings under the **Window settings** and **Video settings** heading to control the input and output of the AVI export.

Window settings

The **Window settings** control the input to the AVI file, i.e. the view that is used for the export. The settings consist of a list of active and saved views (3D or 2D). Select the view that you want to use in an export, then you set the output with the **Video settings**

below. That view will then be saved as **Previous settings** in the list and used in the export until you select another view. It is important to notice that if you make an AVI export from the **File** menu, the new settings will be used in the export directly after a capture. Therefore it is recommended that you save any views with the **Save View** option, that you want to be able to use again.

The list has the following columns:

Name
The name of the view depends on whether it has been saved or not.
The active views, i.e. those views that are open in QTM, are called Window 1 and so on. Then there is a saved view with the last AVI export made by QTM, it is called **Previous settings**. The name of these two cannot be changed and you cannot delete the view from the list.
The views that you have saved with the **Save View** option can have any name. Double-click on the name to change it. The saved views can also be deleted by right-clicking on the view and select **Delete**.

Type
The type is either **3D** or **2D**. A 2D view that only displays video images are still a 2D type of view. Then the type can be either **Active** or **Saved**.
Note: If you are saving an Oqus video to an AVI file, e.g. with a 3D overlay of, then the video image will be linearized. I.e. the same parameters that are used to correct the marker data is applied to the video data to correct for the lens distortion. Therefore the pixels will not match exactly with the original video image. The linearization can be turned of with the **Show linearized data** option on the **2D view settings** page in **Project options**.

Size
The size is the x and y size of the view in pixels.

View count
The **View count** displays the number of cameras displayed in a 2D view. A number within parentheses means that there that number of views with a 3D overlay.

Below the list there are three options.

2D view text
The **2D view text** option toggles whether the text, e.g. camera number, is displayed in the AVI export.

2D view borders
The 2D view borders option toggles whether the borders around a camera view is included in the AVI export.

Save View
The **Save View** option save the selected view so that you can use it later. The view is then copied and the type changed to **Saved**. The name can be changed directly in the **Name** column.

Video settings

The **Video settings** control the output of the AVI export. The following options are available:

Width
The width of the video in number of pixels. The width can also be changed by resizing the **Preview** window.
Note: The width can be larger than the display size.

Height
The height of the video in number of pixels. The width can also be changed by resizing the **Preview** window.
Note: The width can be larger than the display size.

Frame rate
The frame rate in Hz of the video file. The rate is down-sampled if you use a frame rate lower than the marker or video capture in the view. I.e. if you enter 30 Hz and the file is captured at 120 Hz, then the video file will contain every fourth frame.
Note: For just playback on a computer it is sufficient to use 30 Hz.

Playback speed
The **Playback speed** option controls the speed of the file in % of the original speed, so that the file can for example be played in slow motion. E.g. if you have a file captured at 120 Hz and a **Frame rate** for the AVI export of 30 Hz, then you can use a **Playback speed** of 25% to get all of the captured frames in the video file at a quarter of the original speed.

Preset
The **Preset** option is for using standard video output in the 16:9 format. The available presets are:

> **Custom**
> The **Preset** option is set to **Custom** as soon as any of the **Video settings** does not match the other three presets.
>
> **480p - 30 Hz**
> The video output is set to 480p at 30 Hz. This means a size of 854*480 pixels and using progressive scanning.
>
> **720p - 30 Hz**
> The video output is set to 720p at 30 Hz. This means a size of 1280*720 pixels and using progressive scanning.
>
> **1080p - 30 Hz**
> The video output is set to 1080p at 30 Hz. This means a size of 1920*1080 pixels and using progressive scanning.

Use window dimensions
The **Use window dimensions** setting makes the size of the video the same as the window that is selected in the **Windows settings**.

Use window aspect ratio
The **Use window aspect ratio** setting makes the ratio of width and height the same as the window that is selected in the **Windows settings**.

Codec
Use a codec to reduce the file size of the video. Trying to save an uncompressed video will make the file size very large. The two recommended codecs are FFDShow-MJPEG (lossy codec) for standard video that can be played at normal speed and Lagarith (lossless codec) if you need to keep all the video information. The Lagarith codec is usually to slow to be played at normal speed.
There are two options for the codecs:

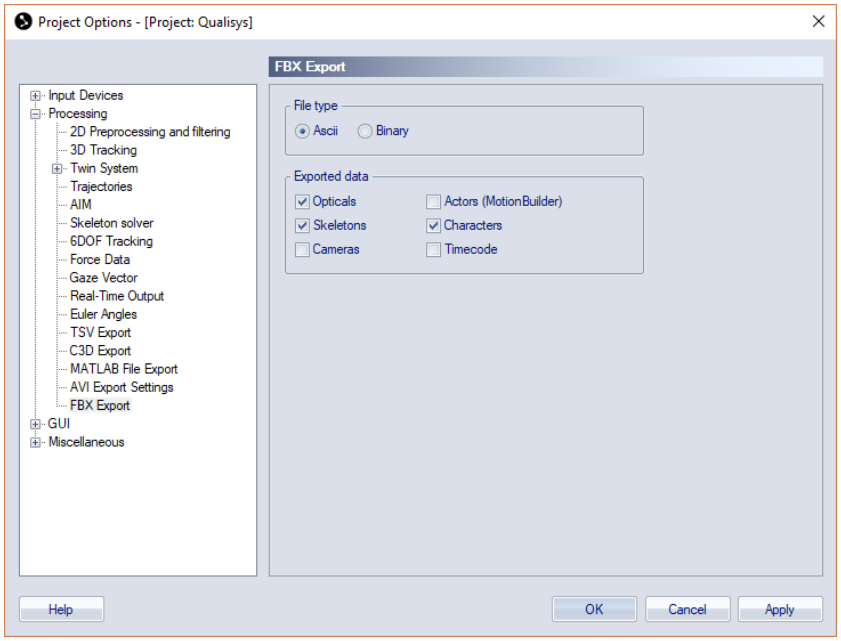
> **Codec settings**
> Click on Codec settings to open the settings for currently selected codec.
>
> **Quality**
> For some codecs you can set the **Quality** directly without opening the settings.

Preview
Display a preview of how the exported window will look like, showing the data of the current frame. The size of the video window can be changed by resizing the preview window. However, if the **Use window dimensions** setting is active then the size is locked.

Marker/Video frequency
The marker and video frequency of the current camera settings are displayed at the bottom of the page. The video frequency that is displayed is only the highest of the used frequencies.
If you are exporting an AVI file from a QTM file then the marker/video frequency displays the frequencies of that file.

The settings of the previous export are always saved, that means that if you change settings for an export from the **File** menu, those settings will also be used for an export directly after a file capture. The settings are not saved if you use the **Export view to AVI** option on the **3D** and **2D view window** menus.

FBX export



On the **FBX export** page there are settings for the FBX export. For information about the FBX export, see chapter “Export to FBX file” on page 406.

The following settings are available for FBX export.

File type
Choose if the exported data is in ASCII format or in binary format.

Exported data

Opticals
Export labeled trajectory data.

Skeletons
Export skeleton data.

The data is scaled according to the **Scale factor** used in the **Skeleton solver** settings of the file. To change the **Scale factor**, you need to reprocess the file

with the modified skeleton solver settings, see chapter “Reprocessing a file” on page 327.

Cameras
Export poses of calibrated cameras.

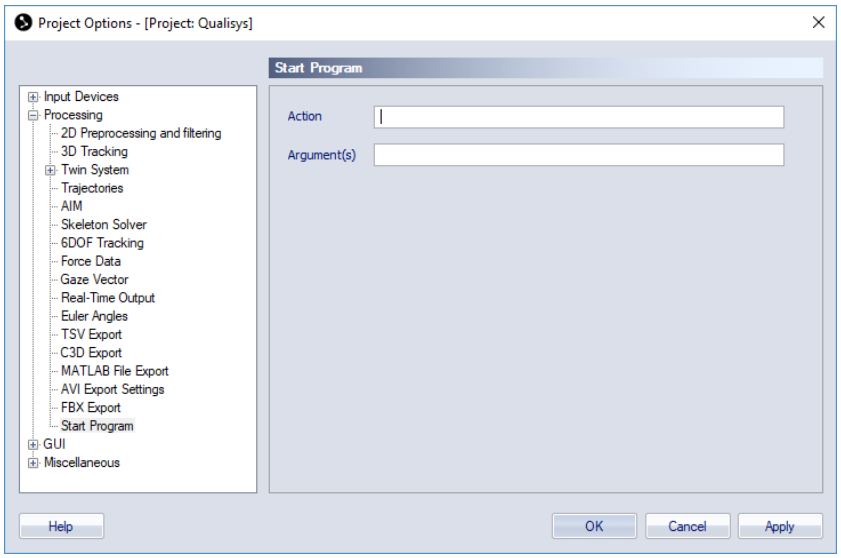
Actors (MotionBuilder)
Export Actor(s) that can be used for IK solving in MotionBuilder. See MotionBuilder documentation for more information. Requires that skeleton data has been calculated and that the **Opticals** option is enabled.

Characters
Export Character(s) that can be used for retargeting in third party software. A Character provides a mapping from QTMs skeleton structure to a predefined structure, defining which segment is the head, the legs, the arms and so on. This in turn can be used to drive 3D models with a slightly different skeleton structure. Requires that the **Skeletons** option is enabled.

The data is scaled according to the **Scale factor** used in the **Skeleton solver** settings of the file. To change the **Scale factor**, you need to reprocess the file with the modified skeleton solver settings, see chapter “Reprocessing a file” on page 327.

Timecode
If enabled, SMPTE timecodes (when available) will be used as timestamps for any exported data (Opticals or Skeletons).

Start program



Start an external program using the command "Action Argument(s)". This can be useful for further automatic processing of an exported file.

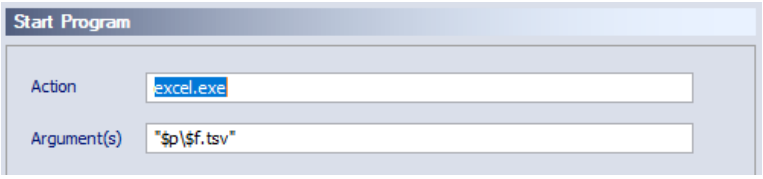
Action
Specify the program to be started.

Argument(s)
Specify additional arguments to be added to the command. The following parameters are available:

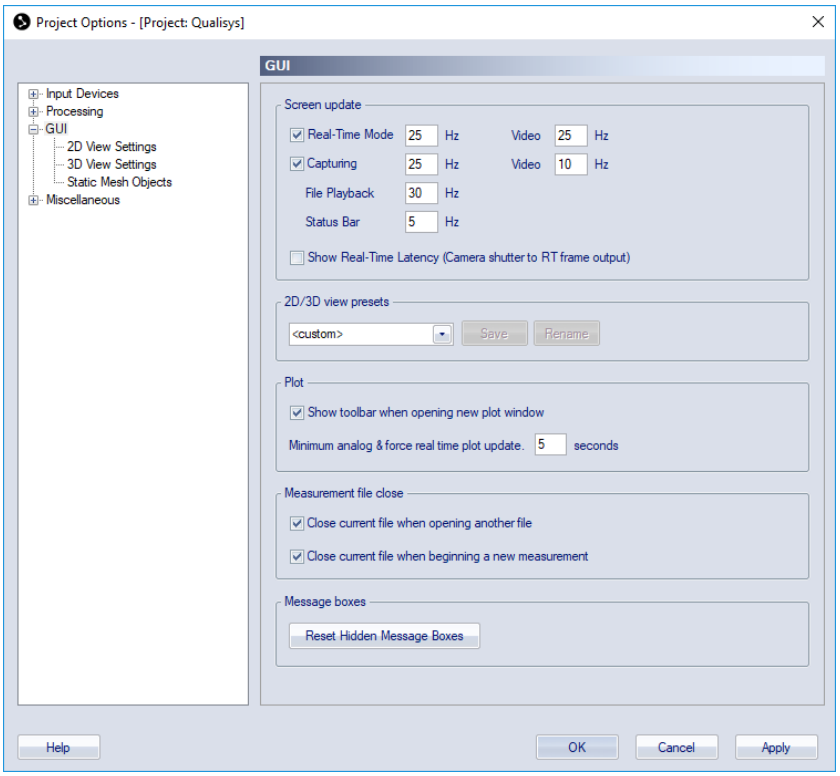
$p: Path of the current capture

$f: File name of the current capture

Example: To open a capture in Excel, make sure that **Export to TSV file** is selected and use the **Action** and **Argument(s)** fields as below.



GUI



The **GUI** page 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**. Under the GUI main page there are pages for **2D View Settings**, **3D View Settings** and **Static Mesh Objects**.

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). Uncheck the check-box next to the setting to disable the GUI during preview. The screen update rate for **Video** can be set independently.

Capturing
Set the frequency for the screen update during capturing. Uncheck the check-box next to the setting to disable the GUI during capture. The screen update rate for **Video** can be set independently.

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 315.

The **2D/3D view presets** options are used to control and save presets for the 2D and 3D GUI.

Preset drop-box
Select the preset from the **Preset** drop-box. There are two standard presets: **QTM default** and **QTM High Contrast**. These can be used to set the default settings respectively to change the colors to improve the visibility in sunlight.
Then there are 8 user presets that can be saved with any 2D view and 3D view settings.

Save
To save a preset select one of the 8 user presets in the **Preset** drop-box and then click on **Save**. All of the settings on the **2D view settings** and **3D view settings** pages are saved to the preset.

Rename
Rename the currently selected preset.

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.

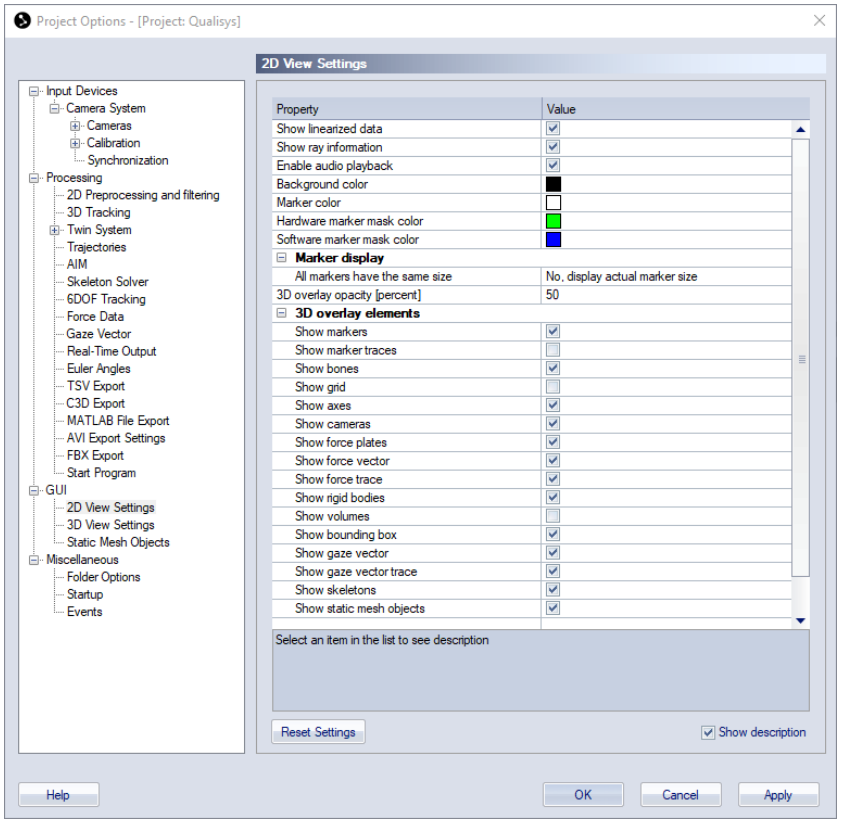
The **Measurement file close** options are used to control how QTM closes files automatically.

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 project and are applied both in the RT/preview and on a opened file. You can 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.

Note: Marker masks and the red rectangle representing the image size are not drawn linearized, this means that with wide angle lens it is best to turn off the **Show linearized data** option to see the true positions of the mask and image size.

Show ray information
Enable the display of the trajectory color for the marker in the 2D views, i.e. display which markers that are used for a trajectory.

Enable audio playback
Enable audio playback for captured video from DV-cam and webcam.

Background color
Select the **Background color** of the marker views.

Marker color
Select the **Marker color** in the 2D views.

Hardware marker mask color
Select the **Hardware marker mask color** in the 2D views.

Software marker mask color
Select the **Software marker mask color** in the 2D views.

Marker display

All markers have the same size
Select whether the 2D markers are displayed with their actual size or with the same size.

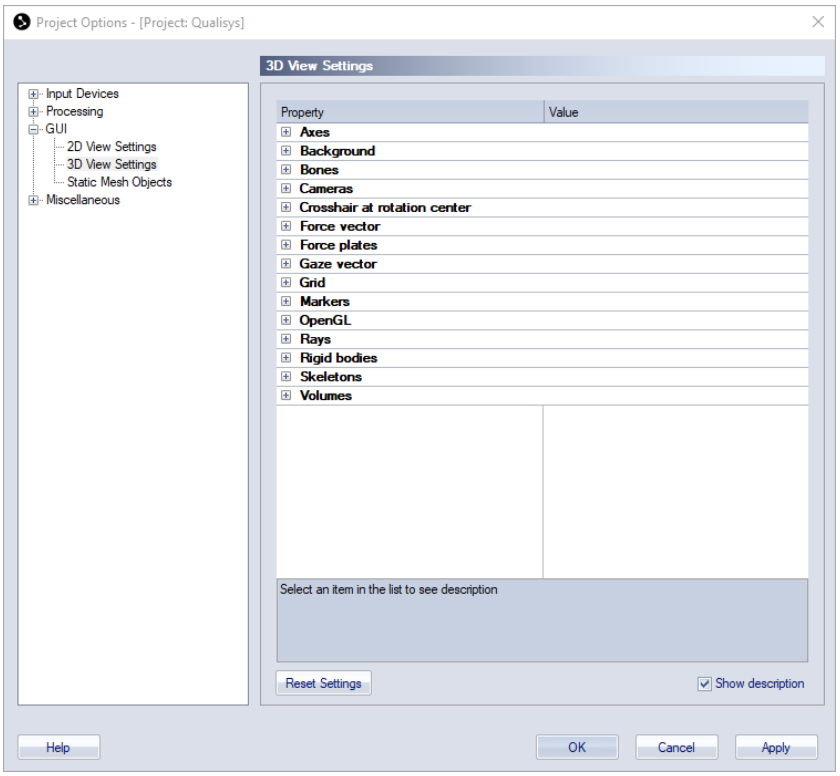
Marker size
Select the marker size in subpixels when **All markers have the same size** is set to **Yes**.

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**, **Gaze vector**, **Gaze vector trace**, **Skeletons**, **Static mesh objects**.

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 project and are applied both in the RT/preview and on a opened file.

You can 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.

Note: The display of some of the 3D view elements can be changed directly on the **GUI Control** toolbar.

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 default 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 vector
Display settings for the force vector in the 3D view.

Show force vector, **Force vector color**, **Scale factor (mm/N)**, **Show force trace**
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**,

Gaze vector
Display settings for the Gaze vector of eye-trackers.

Show gaze vector, **Gaze vector color**, **Gaze vector length**, **Show gaze vector trace** and **Gaze vector trace 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)**, **Number of subdivisions**, **Thicker lines on outside and center**
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**, **Show labeled trajectory information**
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 selected trajectories and the total number of trajectories in the current frame at the bottom right corner of the 3D view window.

The **Show labeled trajectory information** option shows the status of the labeling at the current frame in the upper right corner of the 3D view window.

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.**

Rays
Display settings for camera tracking rays in the 3D view.

Enable camera rays
Show/hide camera rays

Length of ray excess [mm]
Additional length of ray beyond the calculated intersection point.

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**, **Show rigid body wireframe**, **Show rigid body mesh**
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.

Skeletons
Display settings for skeletons in the 3D view.

Show skeletons, **Show skeleton labels**, **Skeleton color**, **Skeleton thickness**, **Show segment labels**, **Show segment coordination axes**, **Show segment markers**

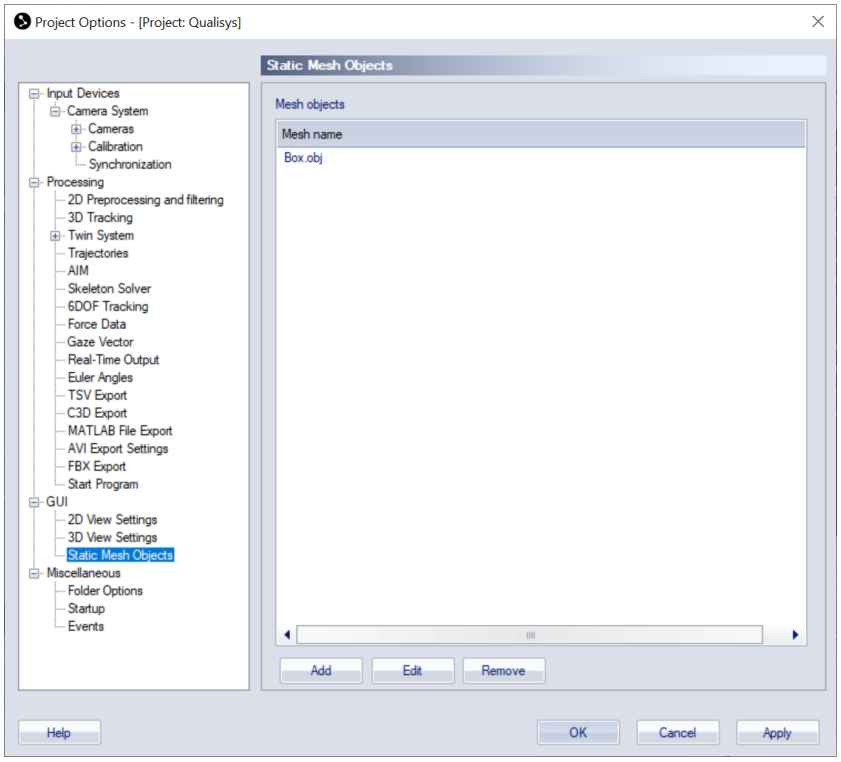
Static mesh objects
Display settings for static mesh objects in the 3D view.

Show static mesh objects

Volumes
Display settings for the display of volumes in the 3D view, for more information see chapter “Volumes in 3D views” on page 54 and “Camera view cones in 3D views” on page 53.

Show covered volume, **Cut covered volume at floor level**, **Cameras required to consider volume covered**, **Show calibrated volume**, **Enable camera view cones**, **Length of camera view cones [m]**, **Show bounding box**

Static mesh objects



The **Static Mesh Objects** page contains a list of static 3D mesh objects that are shown in the **3D view window** in the project. Mesh objects are Wavefront 3D object files.

Static meshes in this list are associated with and saved in the project. The 3D meshes are not stored in QTM files, which means that the whole project must be shared to include the meshes when opening a file.

The static meshes for a project will be shown in the **3D view window** independent of the file or measurement being displayed. Use the **Show static mesh objects**option on the **3D View Settings** page to disable the display.

Note: Using really large meshes can slow down the rendering in QTM.

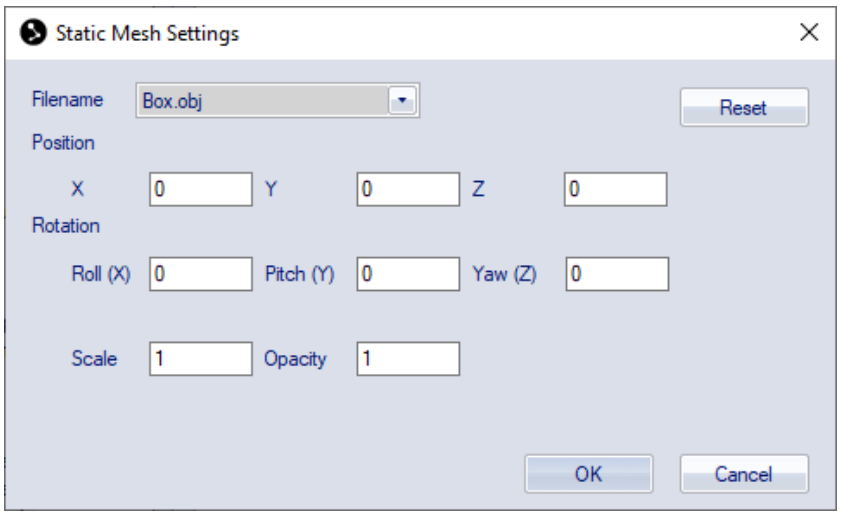
The buttons can be used to manage the list or access and edit the settings for the objects:

Add
Add a new object to the **Mesh Objects** list. Choose an .obj file in the **Static Mesh Settings** dialog that opens and set the properties. The mesh is always scaled so that the longest side is 1 m, because there is no standard for how to save the size in the .obj file.

Edit
Open the **Static Mesh Settings** dialog for inspecting or modifying the **Mesh Object** settings.

Remove
Remove the selected mesh object from the list.

Static Mesh Settings dialog



The **Static Mesh Settings** dialog is used to inspect and modify the static mesh object settings.

The dialog contains the following settings:

Filename
A drop-down list of mesh objects in the *Meshes* folder of the project. Copy the .obj file, and any associated .mtl or image file, manually to the *Meshes* folder to use the mesh in the project. The path to the *Meshes* folder can be set on the Folder options page, see chapter “Folder options” on page 250. For information about obj files and the features supported see chapter “Compatibility of meshes” on page 206.

Position
The 3D translation of the object relative to the global coordinate system. The values of **X**, **Y** and **Z** are specified in mm units.

Rotation
The 3D rotation of the object relative to the global coordinate system. The values for **Roll**, **Pitch** and **Yaw** are specified in degrees around the X, Y and Z axes, respectively.

Scale
The scale factor of the mesh object.

Opacity
The opacity of the mesh object from 0 (transparent) to 1 (opaque). An .obj file can include an opacity option for individual faces, these will be rendered as transparent even if the **Opacity** option is set to 1.

The buttons:

Reset
Remove the link to the file and resets the **Position**, **Rotation** and **Scale** values.

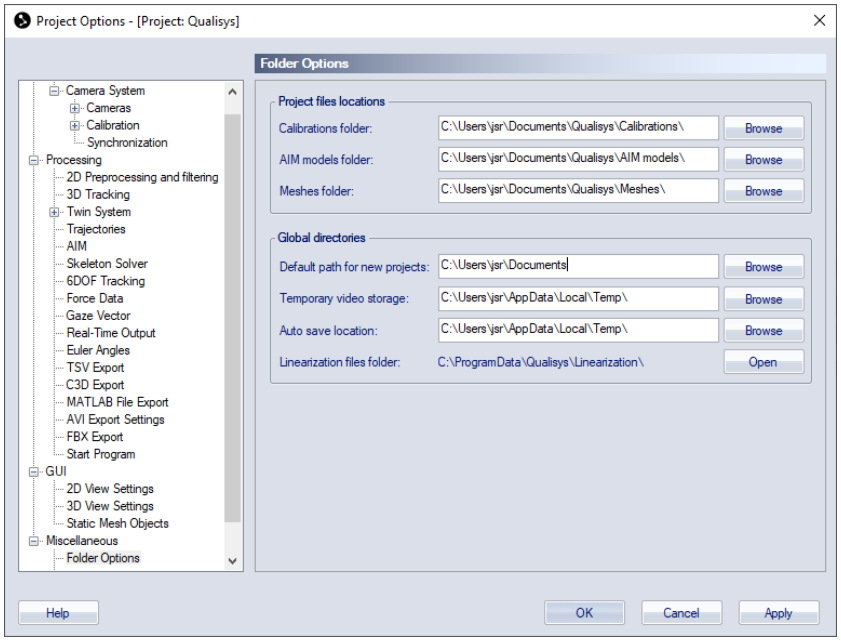
OK
Accept the settings and close the dialog.

Cancel
Discard the settings and close the dialog.

Miscellaneous

The **Miscellaneous** tree contains the settings pages for**Folder options**, **Startup** and **Events**.

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 to 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.

Meshes folder
The **Meshes folder** is by default set to a folder called Meshes in the Project folder. It can be changed to another folder with the **Browse** option, for example if you want to use the same meshes tin different projects. However, then you must change it manually in each project.

The **Global directories** are used by all projects.

Default path for new projects
The default path is used when you create a new project. Change it with the **Browse** option, if you do not want the Documents folder to be the default folder to save the projects in.

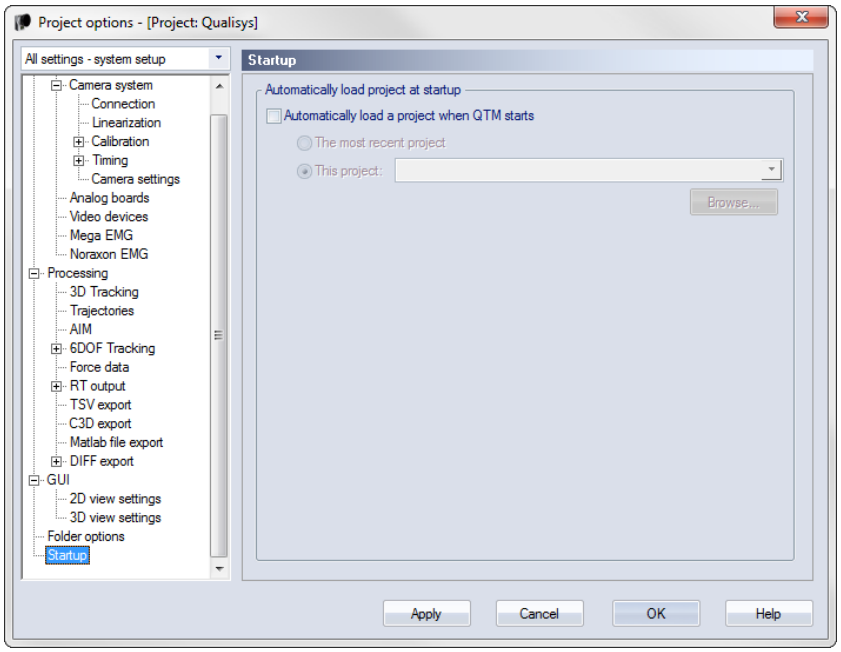
Temporary video storage
The **Temporary video storage** is used for storage of the temporary video files of

Qualisys video and external video 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.

Auto save location
The auto save location is used for storage of the temporary auto saved captures in case **Auto backup** is enabled under Capture actions on the **Processing** page. 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



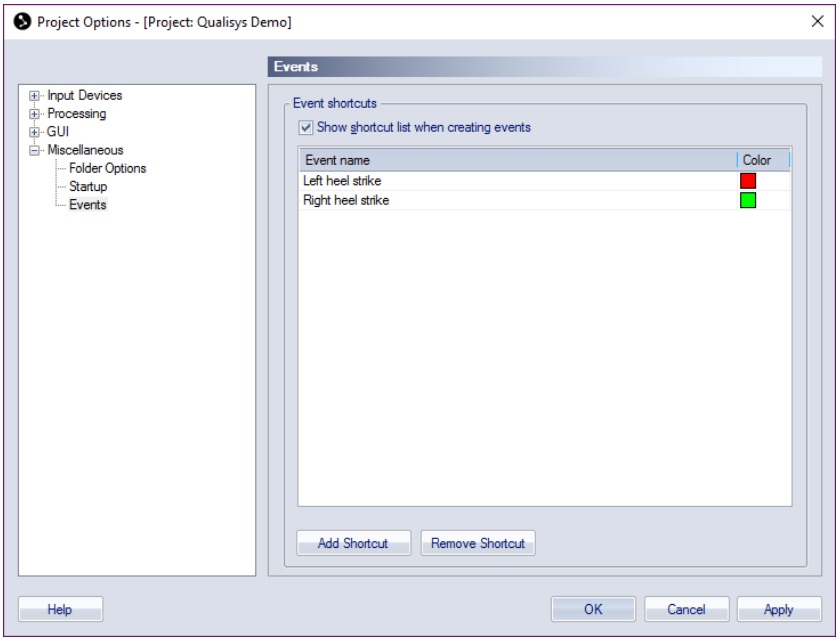
The options on the **Startup** page changes how projects are loaded when QTM starts. The default is that no project is loaded automatically and that you have to select a project from the **Open project** dialog, see chapter “Manage projects” on page 18.

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.

Events



The **Events** page contains settings for **Event shortcuts**. For information on how to set and edit events see chapter “How to use events” on page 383.

Event shortcuts

The **Event shortcuts** options are for creating default event names that can be easily accessed in the **Add events** dialog when creating an event, see chapter “Adding events” on page 383. The shortcuts are enabled in the dialog with the option **Show shortcut list when creating events**.

Use the **Add Shortcut** and **Remove Shortcut** buttons to add/remove shortcuts to/from the list. The **Event name** and **Color** can be edited in the list.

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” on page 424.

The connection of the camera system to the QTM software is performed on the **Camera System** page in the **Project Options** dialog, see chapter “Locate system” on page 124. 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 443.

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.

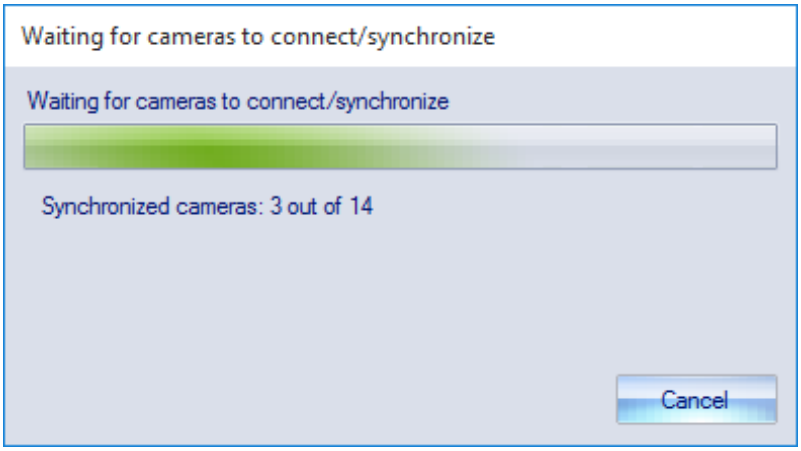
Follow these steps to connect the camera system to QTM:

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 **Camera System** page.
3. Click **Locate System**.
4. Click **Start**.
5. Choose the camera system and click **OK**.
6. For Miqus cameras there is the option to automatically order the cameras using the **Auto Order** button.

Note: In case there are problems to connect the camera system, check the troubleshooting list in chapter “Troubleshooting connection” on page 491.

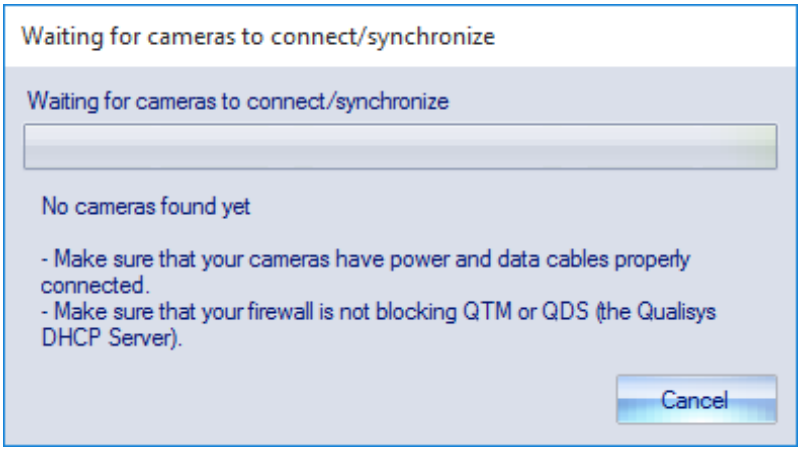
Waiting for cameras to synchronize after new file

QTM will wait for Qualisys cameras to be connected and synchronized when opening a new file with **New** in the **File** menu. The dialog below is shown until all of the cameras are synchronized.



If you start a new file before the cameras have started, QTM will start waiting for the cameras to startup and synchronize, see dialog below.

Note: In case not all cameras have finished booting QTM may not find all cameras. It is recommended to use the connect the cameras by locating the system first, see chapter “Locate system” on page 124.



Optimizing the camera system and settings

The following chapters contains guidelines on how to best position the cameras, how to optimize the marker settings and the usage of the IR markers.

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 269.

  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 276.
- 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.

Identifying and ordering the cameras in QTM

When the camera system is connected the cameras appear in QTM in arbitrary order when starting a measurement for the first time. It can be helpful to order the cameras in QTM so that the order corresponds to the physical setup. There are several ways to achieve this.

Firstly, you will need to identify the cameras. The way cameras can be identified depends on the camera type.

- Arqus and Oqus cameras show their camera number in QTM on the front display.
- Arqus and Miqus cameras can be identified using the **Identification tool**, which can be activated from the **2D View toolbar**. For detailed information, see chapter “Identifying the cameras” on the next page.
- Another pragmatic way to identify a camera is to go to and point at it while watching the live camera video feeds in the 2D view window. This is easiest when you are with two persons.

Ordering the cameras can be done in two ways. For Miqus systems you can use the Auto Order option when connecting the system. This option is only supported for Miqus systems. For more information, see chapter “Automatic ordering of the cameras (Miqus)” below.

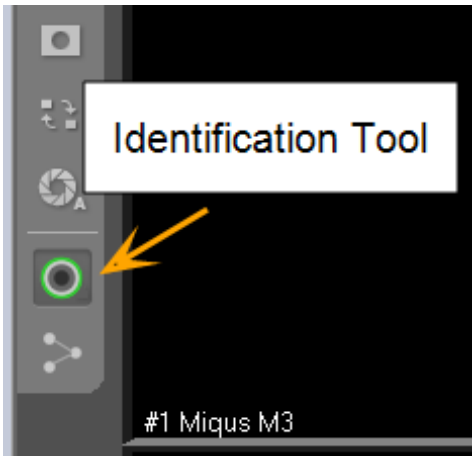
It is also possible to order the cameras manually using the **Reorder** tool in the **2D view toolbar**. Follow these steps:

1. Start a new measurement in QTM and open the **2D view window**.
2. Make sure that all cameras are selected in the **Camera selection** toolbar.
3. Click on the **Reorder tool** on the 2D view toolbar to activate it.
4. You can now drag and drop individual camera image areas to their desired positions. Repeat this until you are satisfied with the order of the cameras.

Identifying the cameras

Arqus and Miqus cameras can be identified/located by using the **Identification Tool** in the **2D view toolbar**. When the **Identification Tool** is active, the LED ring of the cameras selected in the 2D view window will light green. Arqus and Miqus cameras can be identified as follows.

1. Start a new measurement in QTM and open the **2D view window**.



2. Click on the **Identification Tool** icon to activate it.
3. Select one or more cameras using the **Camera selection bar**. The LED ring of the selected camera(s) will light green.

   Note: If you have multiple cameras selected, you can select a single camera by double clicking on its image area. When you double click again, the previous selection is restored.

Automatic ordering of the cameras (Miqus)

The Miqus cameras can be automatically ordered in QTM when locating the camera system. For automatically ordering the cameras, follow these steps:

1. Locate the camera system (**Locate System** button under **Project options** > **Input devices** > **Camera System** > **Connection**).
2. When all cameras have been detected, press the **Auto Order** button in the **Finding Camera System** dialog.
3. When the auto ordering is finished, the **Auto Order** button changes name to **Reverse Order**. The green LED rings of the cameras will flash in the order of

the found sequence. The first camera will light continuously. By pressing **Reverse Order** you can change the order of the cameras, i.e., the last camera of the sequence becomes the first, etc.

Tips on setting aperture and focus

It is very important to set the aperture and focus correctly for your measurement volume. Incorrect settings might lead to sub-optimal marker detection by the cameras.

Aperture and focus is set on the lens of the camera. The way aperture and focus are set depends on the camera and lens combination.

For cameras with motorized lenses (Arqus A12 and Oqus 7+ with standard lens option), the focus and aperture are controlled from QTM via the **Lens Control** interface in the **Camera Settings** sidebar in the 2D View window, see "Camera settings sidebar" on page 31.

For cameras with manual lenses, you will need to extend the strobe mechanics to get access to the focus and aperture rings on the lens. For information on how to extend the strobe mechanics for the available camera types, see the information on setting aperture and focus for the respective cameras in chapter "Setting up the system" on page 424.

It is recommended to use the following procedure for setting the aperture and focus.

1. Select the camera in the 2D view window in QTM using the **Camera selection** bar or by double clicking on the image area.
2. Place a marker in the measurement volume at the position which needs to be in focus. Use the same marker size as used for the actual measurements.
3. Set the focus. The focus can be best set in **Marker** or in **Video** mode. It is best to set the aperture f-value as low as possible (e.g. f/2) when focusing since this makes differences in focus more visible.
   a. In **Marker** mode, make sure that you can only see the marker that you have placed in the volume. Then open a **Data info** window and plot the x-size of the marker. Change the focus until the x-size is as small as possible.
   b. In **Video** mode, change the focus until the marker is as small and as sharp as possible. Make sure that the **Flash time** is long enough in the Camera settings sidebar. You can also zoom in and translate the 2D view for better detail.
4. Set the aperture. The recommended range of aperture is between f/2 and f/8. The optimal setting depends on the following considerations.
   - For more incident light, for example in large capture volumes, use lower f-values from f/2 to f/2.8.
   - For more focal depth, use higher f-values from f/4 to f/8. Use of higher f-values can be helpful in smaller volumes where the relative distance between markers and camera can vary, and there is enough light reflected from the markers.
   - Depending on the lens, use of the lowest f-values may lead to vignetting, which may become visible as a deformation of the markers in the vicinity of the edges and corners on the sensor.
   - Note that by increasing the aperture by one f-stop (e.g., from f/2.8 to f/4) the intensity of the incident light is reduced by half.

5. When you have set the focus and aperture correctly, switch to **Marker intensity** mode in QTM to optimize the marker settings **Exposure & Flash time** and **Marker Threshold**. For more information, see “Tips on marker settings in QTM” below.
6. For cameras with manual lenses, do not forget to return the strobe mechanics to the original position.

For cameras with motorized lenses you may consider to lock the aperture and focus to the set values. This can be done using the Qualisys Firmware Installer (QFI) by setting the Lens control option under **Advanced** settings to **Lock settings**, see “Appendix B: Qualisys Firmware Installer (QFI)”. By locking lens control aperture and focus will be locked to the current settings and the Lens control interface will no longer be available in QTM. This can be helpful in fixed camera setups in which focus and aperture need to be constant.

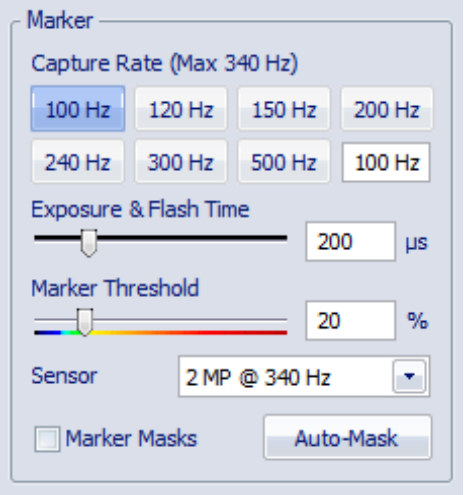
Tips on marker settings in QTM

The two important settings for marker calculation are **Exposure time** and **Marker threshold**. The steps and suggestions below provide a basic method for adjusting the settings. Please note that it is not possible to give more specific advice on how to adjust the settings, because of their dependence on the context and their interrelation. 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 “Tips on setting aperture and focus” on the previous page.

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** at 200-300 microseconds (400 µs for older Oqus series 1, 3 and 5) and **Marker Threshold** to 20. These are the default values and often it is a good starting point.
3. Start by 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 292.

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 31.
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 100 µs (200 µs for older Oqus series 1, 3 and 5) and lower the threshold needs to be low, usually around 10-15.
   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 31.
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 of exposure delay is that the shift in exposure time can cause tracking issues, especially on fast moving markers. The tracker and the calibration compensates for this shift so that it is most of the time close to the quality without exposure delay. However it is still good 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.

1. 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.
2. When you know the cause of the reflections you place the cameras in different groups on the **Cameras** page in the **Project options** dialog, see chapter “Exposure delay” on page 133.
   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. It is also possible to change the exposure group of a camera by right-clicking on the camera live feed in the **2D view** window and selecting the exposure group via the context menu.
   e. When the exposure delay is activated **Using delayed exposure** is displayed in the status bar.
3. 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.

The compensation for the 2D position is also used in the calibration, so usually the results will be similar to calibration without exposure delay. However if the wand is moved too fast then the compensation will not work. There is a warning in the **Calibration results** dialog if the wand speed is too high. The speed that you need to use is rather slow compared to what can be used for normal calibrations.

Note: Delayed exposure does not work on the 5-series camera. This is because it 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.

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

QTM can be used with 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” on the next page.

Passive vs Active markers

The difference between passive and active markers is that the passive marker reflects IR-light, while the active marker transmits IR-light. Because the passive marker only reflects light it can be made very light-weight and in many different sizes, therefore it is usually the best option. However in some setups it can be good to have an active marker, for example because of unwanted reflections or at very long distances.

QTM supports these types of active markers:

Short range active marker
This active marker is about the size of a 12 mm passive marker. It has sequential coding controlled by the camera strobe and can also be mixed with passive markers. for more information see chapter “Short range active marker” on page 445.

Long range spherical active markers or **Reference marker**
This large active marker is usually used in industrial measurements, for example in marine measurements. The marker is synchronized with the cameras via a pulsed strobe signal. The same signal is used for spherical and reference marker. Because of the synchronization the marker can be lit only during the exposure time, which means that the LEDs can be brighter. For more information about the long range marker please refer to the information included with the marker.

Untriggered active marker
This type is for any active marker that has a constant light. It therefore does not need any strobe light and the strobe is turned off to minimize unwanted markers. Usually these are markers made by the customer.

Standard marker sizes

The sizes of standard passive lightweight markers are:

Size	Shape
2.5 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. 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.

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. For instructions how to linearize see chapter "Linearize a camera" below.

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. The files are stored in the camera memory. The linearization files of the current system are specified on the **Linearization** page in the **Project options** dialog, see chapter "Linearization" on page 141. On the **Linearization** page you can also deactivate cameras in the camera system.

Linearization is performed at the factory. The linearization file is stored on the camera and downloaded automatically when connecting for the first time to QTM.

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 a Qualisys camera 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 200 subpixels.
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.

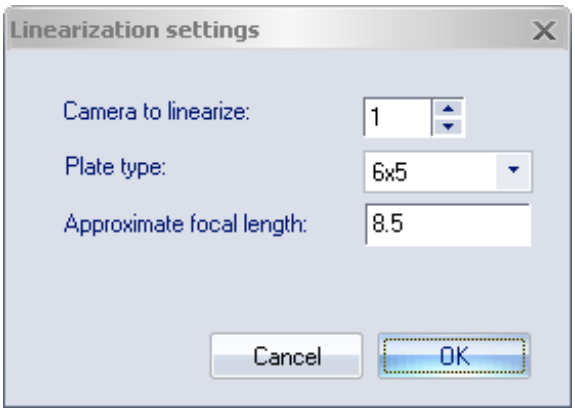
Linearization instructions

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. Set the aperture to a value of 2.8 to 4. You can use higher aperture values of you need a larger focal depth.

   Note: For **Oqus 3- and 5-series:** the recommended aperture is 2.8.
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.
7. **IMPORTANT:** You might need to take off rings or other shiny objects when you perform the linearization. If needed, 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.
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.

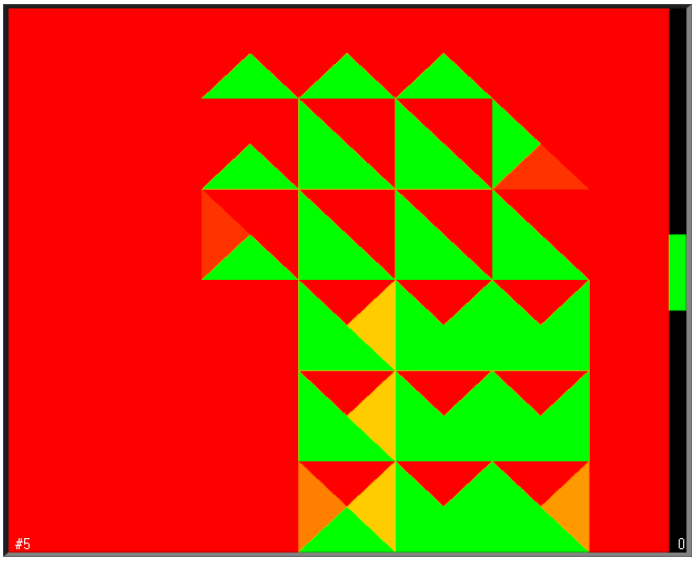
    Oqus 1-series: The standard lens is 8.5 mm.

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

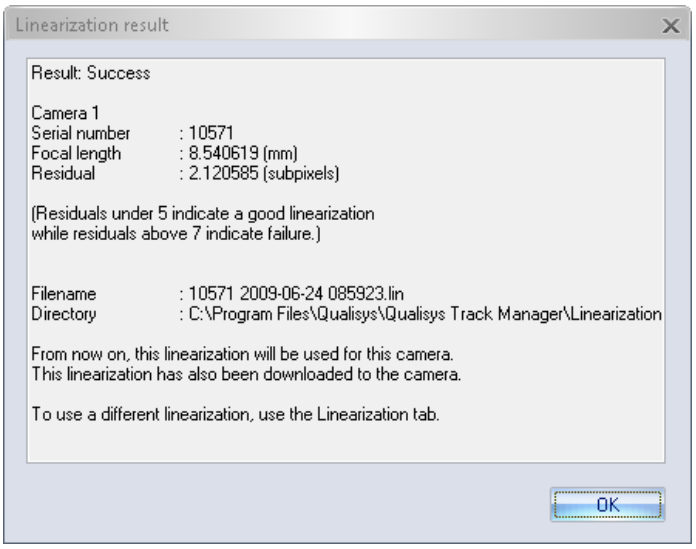
    Oqus 3-series, 3+-series, 4-series and Oqus 5-series: The standard lens is 25 mm (16 mm for 4-series).

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.
    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 longer **Exposure time**.
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.

Note: For wide angle lenses residuals larger than 5 can be acceptable.

17. When you click OK in the dialog the linearization is stored on the **Linearization** page in **Project options**. 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.

    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.

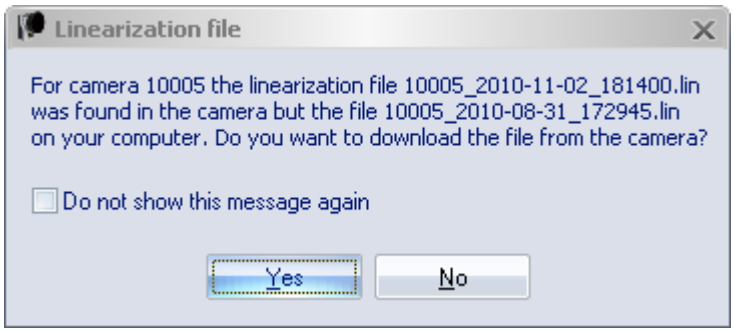
    .qcl - This file contains the actual linearization measurement. This is only named with date and time.

    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 settings and other projects it is a good idea to load the settings, change the lin files and resave the settings and just open the projects and load the new files.

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 two methods that can be used to calibrate a camera system: Wand 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 144. 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 254. 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 273.

A calibration starts with the **Calibration** dialog, see chapter "Calibration dialog" on the 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 23. 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 327.

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:

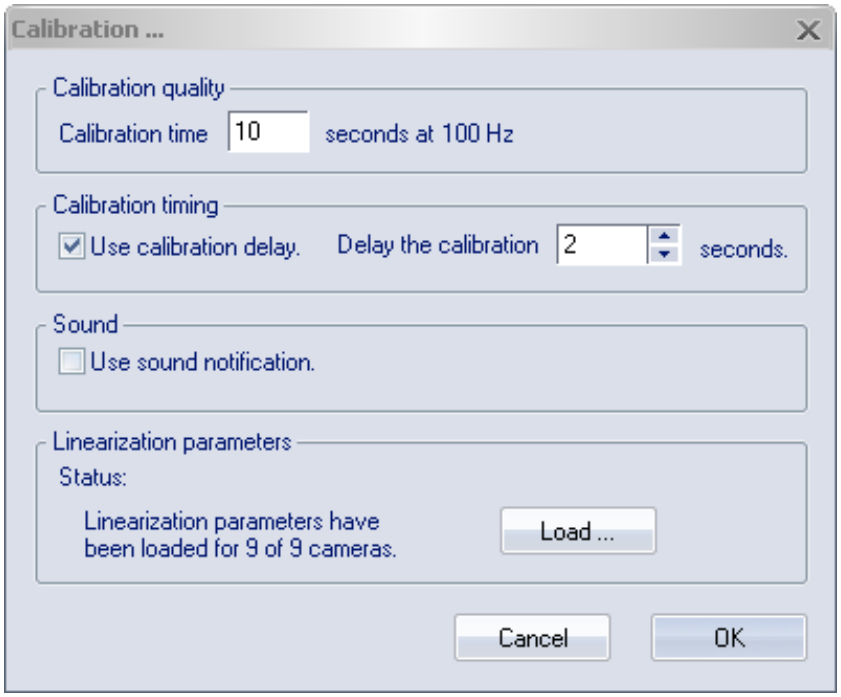
1. Switch on the camera system and start QTM.
2. Open a new file by clicking the **New file** icon.
3. Place the L-shaped reference structure in the measurement volume, see chapter "Wand calibration method" on page 275.
4. Set the settings on the **Calibration** page in the **Project options** dialog, see chapter "Calibration" on page 144.

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 275.
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 493.

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 a Qualisys camera is set to an exposure time longer than 1 ms.

10 Hz - For Qualisys underwater 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

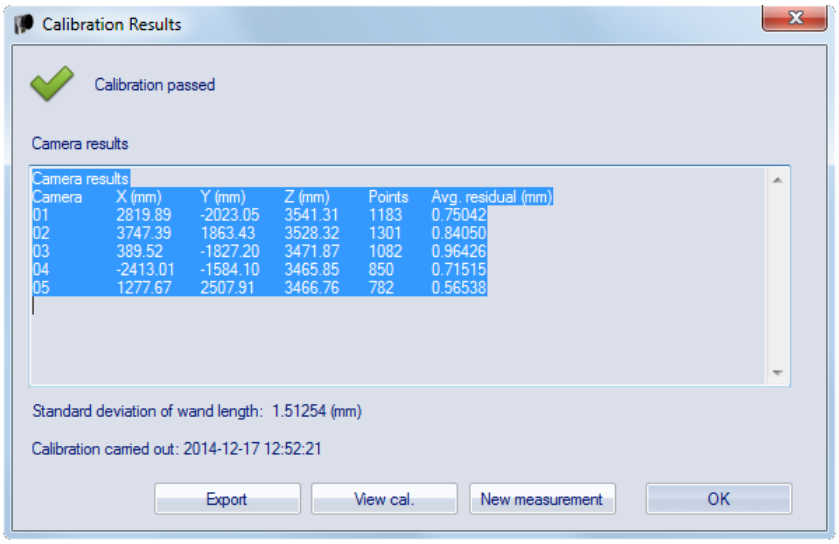
Use the **Sound option** to choose whether to play a sound when the calibration starts and ends.

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 273.



At the top you get a message if the calibration passed or not. There is also a warning if the cameras are using Exposure delay

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 measurement
Close the dialog and the calibration file and open a new capture file in preview mode.

View 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 23.

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.
Note: For wand calibrations the default origin of the coordinate system is in the center of the corner marker, for information on how to translate the origin see chapter "Translating origin to the floor" on page 278.

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 279.

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.

View cal.

With **View 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.

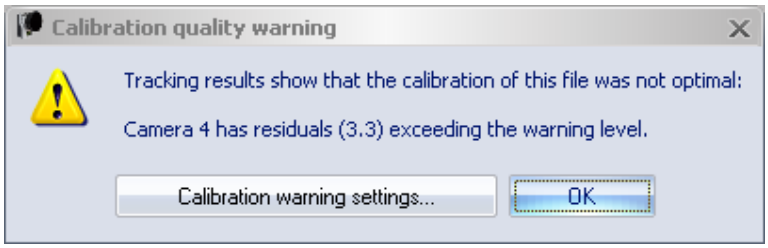
The calibration is new and ok.

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.

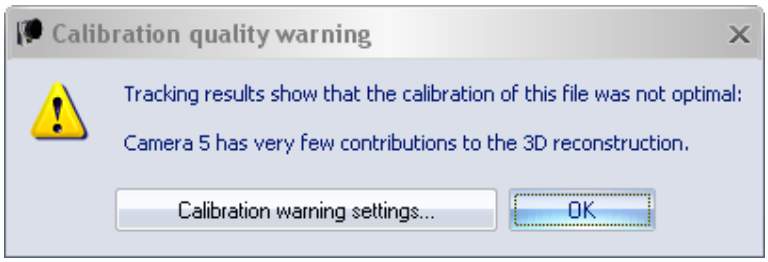


For residuals that are just slightly over the limit the data will still be ok, especially if you have more than 2 cameras that can see each marker. However it is recommended that you check the setup to see if it can be improved and then calibrate the system.

- 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. The standard calibration kit is the 600 mm carbon fiber wand kit described in chapter “Carbon fiber wand kit (600 mm)” on page 278. The 300 mm carbon fiber or 120 mm wand kit can be used for smaller capture volumes.

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 the 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” below.

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.

It is possible to refine the calibration without the L-frame. This method can be used in setups where it is hard to place the L-frame. For more information see chapter “Refine calibration” on page 277.

Calibration tips

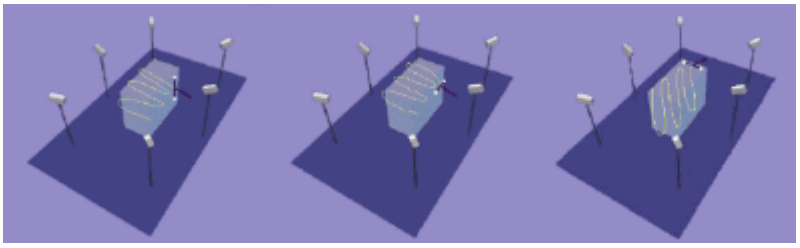
During the calibration, the wand is preferably moved in the measurement volume in a way that allows all cameras to see the wand in as many orientations as possible. This way the cameras will be properly calibrated in all directions.

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.

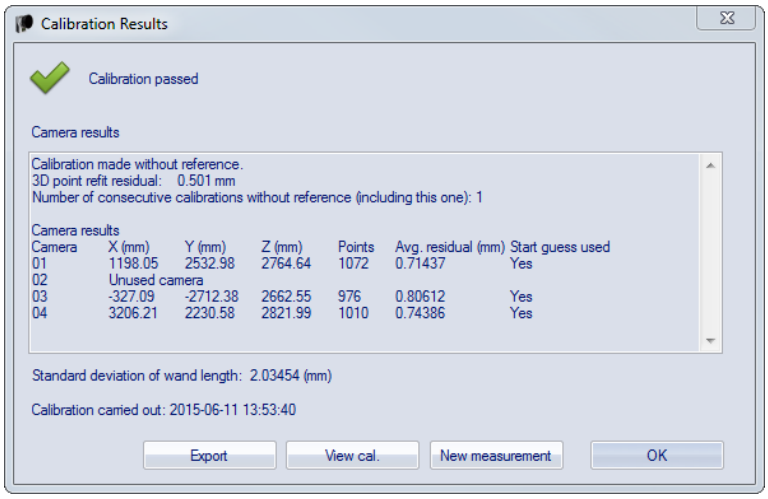
In addition to these volumes of higher importance, it is still important that all cameras can see the wand in as many orientations and positions as possible, just as in a normal calibration.

Refine calibration

Refine calibration is meant to be used in setups where it is hard to place the L-frame. The calibration is then done with only a wand, this however means that coordinate system rotation and position can change since there is no reference for the coordinate system. If the position of the coordinate system is critical then it is highly recommended that you either have other markers that can define the coordinate system after refine calibration or that you use the calibration with L-frame. There is no automatized way to use other markers to define the coordinate system so you need to calculate the rotation and translation in another program and then use the **Transformation** page to update the current calibration.

It is required that the wand can be tracked in the old calibration to be able to refine the calibration. This is because the old wand positions are used to translate and rotate the coordinate system to the best position. This means that if all cameras have moved a lot then refine is not possible to use. However just one cameras or a few cameras can be moved to a completely new position as long as the wand can be tracked and refine will still work.

To use refine calibration start preview and then select **Refine calibration** on the **Capture** menu. Refine calibration uses the same settings as a standard calibration, but in the result you get some extra information.



Extra results for refine calibration

3D point refit residual
The refit residual is a value on how well the new positions of the wand matches the wand positions using the previous calibration. It is impossible to give exact values of when the residual is low enough, since it depends on the volume that is calibrated. However if the Refit residual is higher than 1 mm, then there is a warning in the results because it is likely that the position or rotation has changed.

Number of consecutive calibrations without reference
The the number of calibrations that has been done without an L-frame. The changes in positions of the coordinate system will increase with each new refine calibration. It is not recommended to do more than 5 without an L-frame, unless you have other markers that are used to define the coordinate system in reprocessing.

Start guess used
This displays whether the old calibration position was used as the start guess. For example it will be No if you have moved a camera.

Carbon fiber wand kit (600 mm)

The carbon fiber wand kit consists of two carbon fiber wands with a length of 300 and 600 mm and a L-frame where the long arm is about 600 mm long.

The wand is attached to the handle by pressing and turning it in the track until the handle locks. Make sure that you lock the handle so that the wand is not dropped and the markers damaged.

The L-frame rests on three points. A static corner point that is the origin when calibrating in QTM. The resting points on the arms are adjustable with the adjustment wheels. Check the spirit levels on the frame and adjust the points so that the L-frame is level.

There are also force plate positioning plates on the side of the L-frame, so that the L-frame can be placed on the same position on the force plate for every calibration. Loosen the screws to fold down the positioning plates, then tighten the screws before placing the L-frame on the force plate with the positioning plates on the sides of the force plate.

When folded the L-frame is held together by a magnet. To open the L-frame pull the arms and unfold the arms to the maximum position. The arms are then locked, so to fold it pull the red locking sprint away from the center.

Note: The origin of the L-frame is automatically translated to the corner point (X=-10, Y=-10, Z=-49, relative to the corner marker).

Translating origin to the floor

On the non carbon fiber kits, the origin is by default placed in the center of the corner marker on the L-shaped reference structure. If you want to move the origin to another position you can use the settings on the **Transformation** page, see chapter “Transformation” on page 149. Use the following distances if you want to translate the origin to the bottom corner of the reference structure. The distances are for the Z-axis pointing upwards, the X-axis pointing along the long arm and Y-axis pointing along the short arm.

Wand 750

x: -13.5

y: -13.5

z: -24.5

Wand 300

x: -8.5

y: -8.5

z: -20.5

Wand 110

x: -10.0

y: -10.0

z: -7.5

Note: For the 300 and 750 reference, the bottom corner of the L-shape is 1 mm outside of the U-profile. The figures above are from the corner of the bottom plate, if you want the U-profile you should subtract 1 mm from x and y.

Note: For the new wand kits (120, carbon fiber 300 and 600) the origin is automatically translated to the corner point of the L-frame at floor level.

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 147.

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

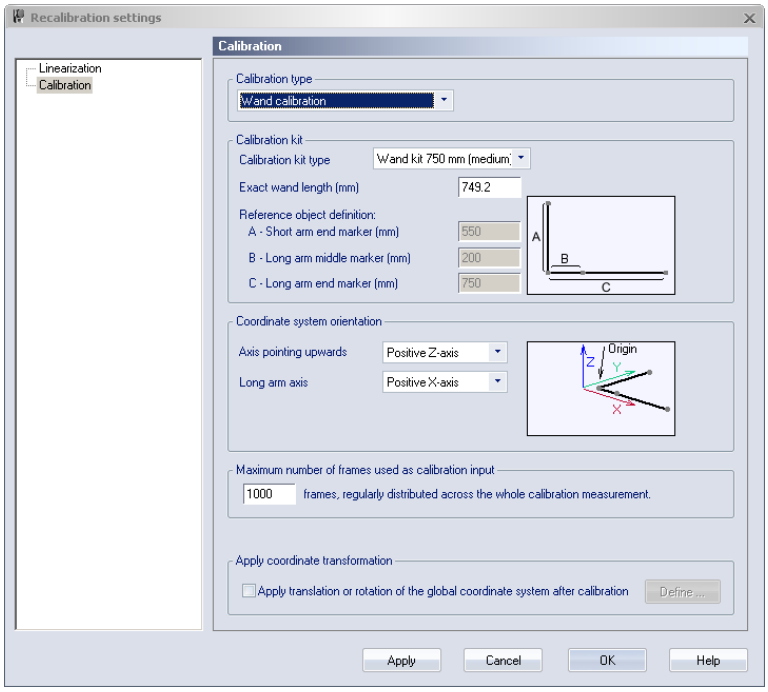
Note: If the camera system has not been moved it is better to make a new calibration and then reprocess the file with that calibration file.

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 23.

   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 144 and chapter “Linearization” on page 141.

   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.

   Note: It is possible to **Use** a calibration that does not match the current camera system, because it enables you to reprocess files easier.

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 327.

Capturing of measurements

Introduction to capture

To capture a measurement you need to have a project in QTM. It is recommended to make a new project if you for example change the marker setups or if you want to work with specific camera settings. For more information about projects see chapter “Projects” on page 14.

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 next 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. When capturing, there will be a red border around the view window.

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 443.

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.

For guidelines on how to optimize the camera system and settings for the best measurement results, see chapter “Optimizing the camera system and settings” on page 254.

Outline of how to capture

The steps below provide an outline of how to perform a capture.

1. Switch on the camera system and start QTM.
2. Select the project you want to use or create a new project.
3. 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.
4. Check that the markers are visible, otherwise change the camera settings see chapter “Tips on marker settings in QTM” on page 258.
5. Calibrate the system, see chapter “Outline of how to calibrate (Wand calibration)” on page 269.
6. Go the **Processing** page in the **Project options** dialog to activate any processing steps.

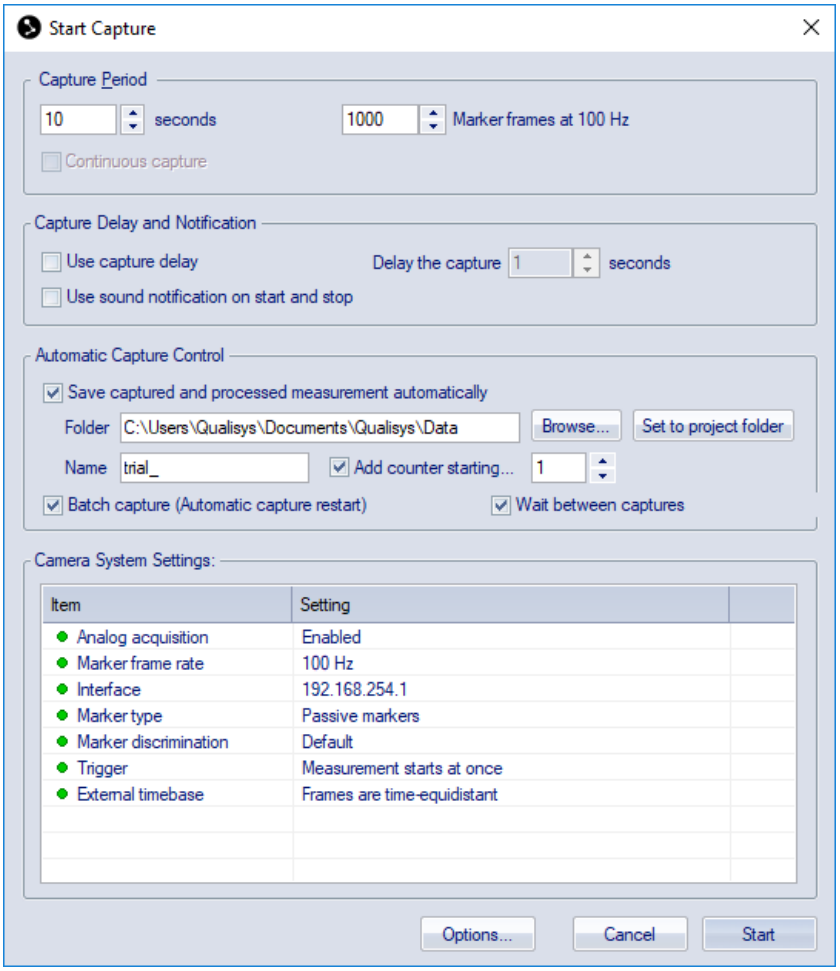
7. Click the **Capture** icon ●.
8. Specify the capture settings in **Start capture** dialog.
9. Check that all of the settings under the **Camera system settings** heading are correct.
10. Click **Start** to start the capture. When the capture is finished, all of the processing steps will be performed and then the new motion capture file is displayed in QTM, unless batch capture is activated. In batch capture, QTM will immediately start waiting for the next capture as soon as the processing of the previous capture is finished. An ongoing capture can always be stopped with **Stop capture** on the **Capture** menu or by clicking the **Stop capture** icon ■.

    Note: To reduce the time between captures enable **Store real-time data** on the **Processing** page, see chapter “Store real-time data” on page 284.

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 494.

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 a Qualisys 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 **Continuous capture** option the capture does not stop until a trigger stop event is received or 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.

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 in the **Folder** option. You can **Browse** for a folder or use the **Reset to project folder** option to reset the folder to Data folder in the current project. Set the name of the files in the **Name** option, 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. When **Wait between captures** is checked the user will be prompted to start each new capture. If unchecked, the next capture will start as soon as QTM is ready after a previous capture. For information on batch capture see chapter “Batch capture” below.

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**.

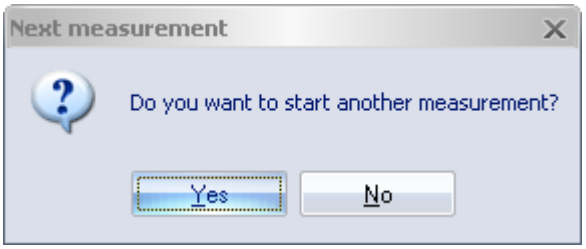
Batch capture

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 **Add 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. During the measurement the border of the view window will indicate the status.

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.

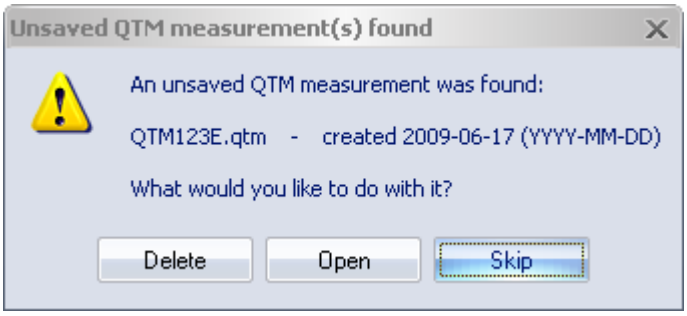
No external trigger
Start each measurement by clicking **Yes** in the **Next measurement** dialog. Stop the batch capture by clicking **No** in the dialog. **Stop capture** on the **Capture** menu can be used during an individual capture to stop just that capture.



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. The **Store real-time data** option can be used to reduce the time between capturesm but still save the data processed in real-time for quick inspection of the data, see chapter “Store real-time data” below.

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.



Store real-time data

The processed real-time data is saved directly to the QTM file when the option **Store real-time data** is enabled on the **Processing** page. This option can for example be used to speed up the capture process and get information if the captured data is good. In this mode only the export processing steps are active and all processing steps that changes data are disabled.

All of the raw data is stored in the file, so that the data can be reprocessed. The data may improve when reprocessing the file, this is especially true for AIM reprocessing which can use all of the data in the file to identify trajectories. The following applies to the different data types:

Data type	Process every frame option enabled	Process every frame option disabled
2D (raw)	All raw 2D data is included in the file	All raw 2D data is included in the file
2D (processed)	All of the 2D data is processed and used for 3D tracking	Some 2D data may not be processed or used for 3D tracking
3D (unidentified)	All of the 3D data is tracked using all 2D data	Some 3D data may not use all of the 2D data available in the file. The result can be 3D data with reduced quality.
3D (identified)	AIM can only use old 3D data to identify the trajectory. When reprocessing the data AIM can use all of the data in the file for identification.	Same as for when the option is enabled and the reduced 3D data quality may also affect AIM .
6DOF	Same as for 3D (identified)	Same as for 3d (identified)
Skeleton	Same as for 3D (identified)	Same as for 3d (identified)
Analog	All of the analog data is included and offsets are applied to all of the analog samples	All of the analog data is included and offsets are applied to all of the analog samples
Force	Forces are calculated for all of the samples	Forces are calculated for all of the samples
EMG	All of the EMG samples are included in the file	All of the EMG samples are included in the file
Eyetracker	Reprocessing is recommended, because the delay between eyetracker and 6DOF data is not compensated for in real-time. The Gaze data is calculated for all of the samples.	Reprocessing is recommended, because the delay between eyetracker and 6DOF data is not compensated for in real-time. Gaze data is calculated for all of the frames that include 6DOF data.
External video	All video frames are included and the time offset is applied to the video.	All video frames are included and the time offset is applied to the video.

Note: When the capture rate is too fast. then the real time frequency is reduced to half of the capture rate or even further reduced. When storing the real-time data it means that 3D data is only processed for all half of the frames, or even less than that for a frequency higher than 400 Hz.

Qualisys video capture

An important feature of Qualisys cameras is the video functionality. All Qualisys cameras can be used in video mode. This can for example be useful when setting up the camera system to see the image area of the individual cameras. Video capture with Qualisys cameras offers the following advantages: 1) synchronized video capture with marker data, and 2) the possibility to use 3D data overlay.

Qualisys offers a range of camera models that can be used as dedicated video cameras. Such cameras feature a clear front glass, a white strobe. Oqus high-speed cameras also have a larger buffer memory buffer for storing video frames. Dedicated video cameras are the Oqus 2c (color) and the Miqus Video (color and monochrome). Furthermore, the Oqus 1, 3, 3+, 5 and 5+ series are available in a high-speed video configuration.

The image resolution and the available range of video capture frequencies depend on the camera model. Some camera models have high-speed sensor modes, which allow video capture at full field-of-view at reduced resolution. Oqus high-speed cameras can capture video at rates up to 10 kHz (uncompressed, reduced FOV). Some camera models feature in-camera MJPEG compression, allowing longer recordings at a reduced maximum capture rate. In standard marker cameras, the video capture rate is limited to 30 Hz. For an overview of features of Qualisys video cameras, see chapters “Oqus high-speed video sensor and memory specifications” on page A - 2 and “Qualisys video sensor specifications (in-camera MJPEG)” on page A - 3.

The following chapters describe how to capture video with Qualisys cameras.

Using Qualisys video cameras

Video capture with Qualisys cameras involves the following steps:

1. Make sure that the cameras that will be used for recording video are in video mode.
2. Choice of video settings (capture rate, exposure, compression, etc.). The available video settings and the interface for changing them in QTM depends on the camera model. The video capture rate can be set individually per camera and may be different from the capture rate of the cameras in marker mode. Some camera models have the option to use in-camera MJPEG compression. This allows a longer capture, however, at a reduced maximum capture rate (see chapter “Qualisys video sensor specifications (in-camera MJPEG)” on page A - 3).
3. Calibration of Qualisys video cameras is optional, but required for 3D overlay. Qualisys video cameras can be calibrated together with the marker cameras in the system. During the calibration, the Qualisys video camera will be automatically switched to marker mode, so it is important to make sure that the marker settings are correct (see “Calibrating Qualisys video cameras” on the facing page). If you do not want to include a video camera in the calibration, you can uncheck it in the linearization options (see chapter “Camera linearization parameters” on page 141).
4. Start a new capture. Depending on the camera model and settings, video is streamed to QTM during the measurement or buffered on the camera and fetched after the capture is finished.

   Note: Depending on the capture frequency streaming video cameras might

store some frames during the capture. These frames are fetched after the capture is finished.

5. The video from each camera will be stored in a separate AVI file and can be played back in QTM (see “Qualisys video files” on page 294). You can then for example activate the 3D overlay on the video (see “3D data overlay on video” on page 295) and export the view with the **Export to AVI** feature on the **File** menu.

Calibrating Qualisys video cameras

Qualisys video cameras are calibrated in the same way as Qualisys marker cameras. During the calibration, the video cameras will be automatically switched to marker mode. Follow these steps to calibrate Qualisys video cameras.

1. First, you need to set the marker settings. Switch to Marker intensity mode and adjust the exposure and threshold so that the markers are red, and the background is dark blue.
2. Make sure that the cameras have enough overlap with other cameras within the system. Preferably, the L-frame is visible, but this is not required.
3. Start a calibration. The video cameras will be calibrated together with the marker cameras in the system.

Note: Qualisys video cameras have a white strobe, because there is a filter that removes the IR light for a better image quality.

Capture streaming video

Streaming video can be recorded with Miqus Video Color/Mono or the Oqus color video camera (Oqus 2c). When using streaming video, compressed video data is sent to QTM during the capture, allowing for long video captures.

The following chapters describe how to capture video with Qualisys cameras.

Miqus Video camera

The Miqus Video camera is a dedicated video camera for capturing synchronized video in QTM. Miqus video cameras can be used with any type of Qualisys camera system, including systems with Miqus Video cameras only. There are two types available: Miqus Video Mono (grayscale image) and Miqus Video Color. The Miqus Video cameras are synchronized in the same way as the other cameras in the system, which means that the video images will always be synchronized with the marker data. It is also calibrated with the rest of the system so that you can use 3D overlay on the video image.

Oqus color video camera

The Oqus color video camera (2c-series) is a dedicated color video camera for the Oqus system that can be used to capture synchronized video. It is synchronized in the same way as the other cameras in the system, which means that the video images will always be synchronized with the marker data. It is also calibrated with the rest of the system so that you can use 3D overlay on the video image.

The Oqus 2c camera can also be used as a high-speed video camera for capturing uncompressed images at higher frequencies. In that case, the video capture duration is limited since the video is stored in the camera's buffer memory during capture. For more information, see chapter “Capture high-speed video” on page 289.

Streaming Video settings

The settings of streaming video cameras (Miqus Video Mono/Color and Oqus 2c) are available under the **Streaming Video** settings in the **Camera settings** sidebar. The **Camera settings** sidebar allows to adjust basic settings, see chapter "Camera settings sidebar" on page 31. Advanced video settings are available on the **Cameras** page in the **Project options**, see chapter "Video settings" on page 134.

The following steps show how you can change the settings for streaming video cameras.

1. Choose the video capture rate. The buttons show a selection of common video capture rates. Integer divisions or multiples of the current marker capture rate are indicated bold. Note that the maximum frequency, as well as the frequency values of the buttons, may change depending on the selected resolution and aspect ratio (steps 2 and 3).
2. Choose the video image resolution. The buttons show standard image resolutions available on the Miqus Video. Note that the field of view (FOV) may change depending on the resolution setting. Both 1080p and 540p utilize the full sensor area, whereas 720p and 480p are achieved by cropping the image area.
3. Choose the aspect ratio. The buttons show standard aspect ratios of 16:9, 4:3 and 1:1.
4. Set the exposure.
    a. When using **Auto Exposure**, you can set the **Exposure Compensation** to get a brighter or darker image. You can also select the area on which the auto exposure is based using the **Auto Exposure Tool** on the **2D view Toolbar** (see "2D view toolbar" on page 29).
    b. When **Auto Exposure** is unchecked, you can manually set the exposure by setting the **Exposure Time** and selecting the **Gain**.

Note: The file size of the recorded videos can become very large when using a high resolution and a high video capture frequency. If you need the video for documentation purposes, you can reduce the file size considerably by using a lower resolution and a lower capture frequency (e.g. 540p @25 Hz).

Note: The option to use custom image sizes is not available for Miqus Video.

Maximum capture rate for streaming video

The maximum capture rate for streaming video cameras depends on the selected resolution and aspect ratio. The available capture frequency range for the current combination of settings is shown under **Video settings** on the **Cameras** page in **Project options**. The below Tables give an overview of the maximum frequency values for Miqus Video and Oqus 2c.

Miqus Video

Maximum video capture frequency (in Hz) as a function of resolution and aspect ratio for Miqus Video.

	16:9	4:3	1:1
1080p	86	111	149
720p	180	237	292
540p	333	428	565
480p	416	550	714

Oqus 2c

Maximum video capture frequency (in Hz) as a function of resolution and aspect ratio for Oqus 2c.

	16:9	4:3	1:1
1080p	24	32	41
720p	53	68	86
540p	62	77	94
480p	77	95	116

Capture video with in-camera MJPEG

The functionality to stream MJPEG compressed video is available on most Qualisys camera models, except the older Oqus camera models. In this mode, the image will be compressed in the camera before it is sent to QTM. This will reduce the amount of data that is sent so that you can measure video for an unlimited amount of time. However, it is important to note that some buffering may occur during the capture and those frames will be fetched after the measurement has ended.

The available capture rates are lower than those for uncompressed video since the camera has to process the image. For example, on the Oqus 2c video camera at 1920*1080 the video capture rate is limited to 24 Hz. It is important to note that there are multiple high-speed sensor modes available on the 2c and 5+ series. These can be used if you want to have a higher capture rate, but want to keep the Full FOV at a reduced resolution, see chapter “Sensor mode” on page 139 and “Qualisys video sensor specifications (in-camera MJPEG)” on page A - 3.

The in-camera MJPEG compression is selected with the Video compression option, either on the **Camera settings** sidebar or on the **Cameras** page in the **Project options** dialog, see chapter “Camera settings sidebar” on page 31 and “Video compression” on page 137. The in-camera MJPEG compression is default when it is available. Miqus Video always uses in-camera MJPEG compression, so it is not available as an option.

You can specify the quality of the MJPEG codec on the **Cameras** page. The default quality is 50, you can increase it if you want higher quality but then the file will be larger.

The video file will be saved with MJPEG compression. Therefore, the computer needs to have codec that can read it to play the file. Mostly, Windows computers can usually play videos compressed with MJPEG codec without having to install a specific codec.

Capture high-speed video

All Oqus cameras with high-speed video capabilities can capture video up to 10000 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 in the **Camera settings** sidebar or on the **Cameras** page in **Project options** dialog, see chapter “Camera settings sidebar” on page 31 respectively “Cameras” on page 127.

Information about high-speed video capture

An 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, see “Oqus high-speed video sensor and memory specifications” on page A - 2.

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, see “Oqus high-speed video sensor and memory specifications” on page A - 2. For another resolution multiply the amount of data with the reduced resolution/full resolution.

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 to at least 4 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 “Synchronization” on page 154.
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.

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 **Cameras** 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.

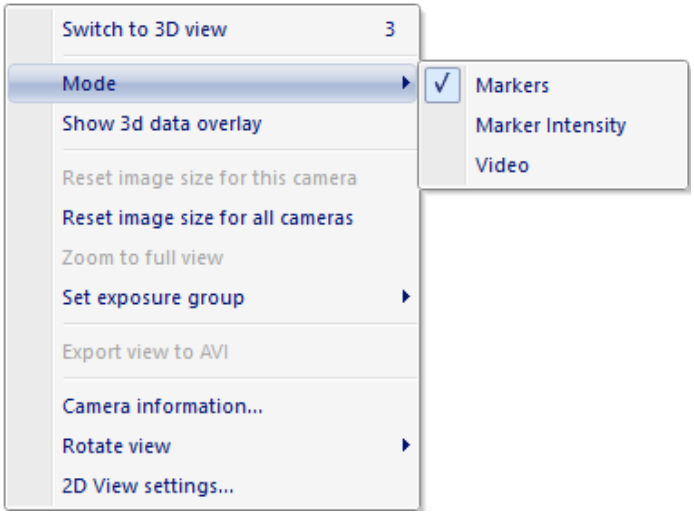
FFDS (MJPEG)

- 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 138.
- In QTM these video files can be played at the original speed. The video files can also most of the time be played on computers even if the codec has not been installed.
- The codec works well with Tema QVA, both playback speed and quality is ok for a lossy codec.
- A tested version of the codec is included in the QTM installer (recommended). The latest version of the codec can be downloaded from http://sourceforge.net/projects/ffdshow-tryout/.

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 Qualisys 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 290.

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 marker cameras will continue to capture at the marker capture rate and try to keep the marker RT frequency. The video frames will then be fetched in parallel with the marker data at the highest possible frequency. Which depends on the camera settings (for example zoom) and the number of cameras in video mode.
   If the camera has In-camera MJPEG compression then that is used during the preview and the RT frequency can be around 20-30 Hz. However for the other cameras the RT frequency will usually be 5-10 Hz.



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 **Cameras** page. This is the mode that you use when you want to capture video with the Qualisys 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 sidebar** to get a bright image in preview with non high-speed Qualisys 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 30.
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.

Qualisys video files

The video captures from each camera will be stored in separate AVI files at the same location as the QTM file. The name of the AVI file(s) is the same as the QTM file with the camera type, camera number and camera serial number appended. Video files are linked to in the QTM file using a relative path, so that the link is still valid when moving or copying all the files to a different location.

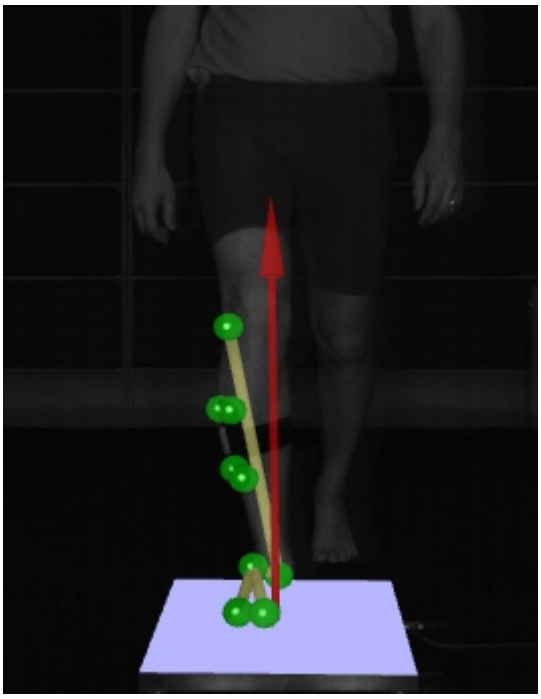
The video data can be compressed with a codec in QTM. You can select any codec that is installed on the computer to compress the data, for information on video codec see chapter “Codecs for Oqus high-speed video files” on page 291. However, for the **In-camera MJPEG** mode the codec is always MJPEG.

The video file is 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 you have compressed the video file with a codec the computer where you play the file must have that codec installed.

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.

3D data overlay on video



The 3D data can be overlayed on the Qualisys 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 Qualisys 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 Qualisys 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 243.
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.

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.

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 259.

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.

Marker masking

How to use marker masking

The marker masking is a tool 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 next page. The masks can also be defined manually on the **Cameras** page in the **Project options** dialog. For explanation of the settings see chapter “Marker masking” on page 131.

Note: Software marker masks can be added to a file and requires reprocessing, see chapter “How to use software marker masks” on page 331

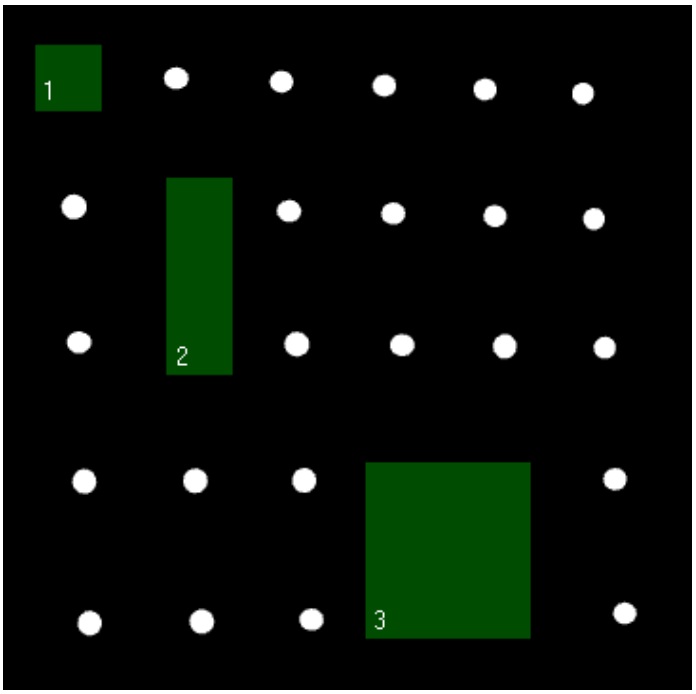
Follow these steps to add a masking area to a camera.

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.

   Note: Marker masks are not drawn linearized, this means that with wide angle lens it is best to turn off the **Show linearized data** option on the **2D view settings** page to see the true positions of the mask and image size.

   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. To delete a mask or all masks for the current camera, right-click on it and select **Delete this mask** or **Delete all masks**.

   Repeat this step until all of the unwanted markers are covered. There can be up to 5 masks per Oqus camera and 20 masks per Miqus 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. The auto mask function will identify the largest reflection and mask them. If there are reflections close to each other, the auto mask function will try to join the masks.

Note: The number of masks per camera is limited to 5 masks for Oqus and 20 masks for Arqus and Miqus. If there are more unwanted reflections you will need to manually edit the 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.

Note: Marker masks are not drawn linearized , this means that with wide angle lens it is best to turn off the **Show linearized data** option on the **2D view settings** page to see the true positions of the mask and image size.

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.

Marker circularity filtering (Oqus)

How to use non-circular marker settings

The non-circularity marker settings can be used to correct or delete bad or partially hidden markers. 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 the non-circular markers. This will result in more 3D data, which otherwise cannot be calculated.

The settings for the Qqus camera is activated on the **Cameras** page in the **Project options** dialog, see chapter “Marker circularity filtering (Oqus)” on page 132. The markers used by the filter are filtered out by the camera depending on the **Circularity level** setting.

How to handle the non-circular markers in then set on the **2D Preprocessing and filtering** page by the two options: **Correct center point of non-circular markers** and **Discard non-circular markers**, see chapter “How to use circularity filter” on page 330.
Note: If you change this option when processing a file, then you must also retrack the 3D data for the change to have any effect.

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.

Red outlines
These are the outlines created from the segments sent from the camera. If there is no marker inside the outline, then it has been discarded. It can only happen with the **Discard non-circular markers** option.

Light Yellow - non-circular markers used by the 3D tracker
These markers are detected as non-circular by the camera, but has not been corrected in the pre-processing since there was no circular fit that was good enough. It can only appear with the **Correct center point of non-circular markers** option.

Green - corrected markers
These markers have been detected as non-circular and then corrected by the filter. The pre-processing tries to fit a circle to the best arc of the red outline.

How to use the marker filtering depends a lot on the camera setup and in some cases using it doesn't help much. Check these points to figure out if it will work in your setup:

Marker size
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 marker size. Therefore all markers below this size are considered as circular markers. Also markers that are too large (>2000) will make it hard for the camera to have time to calculate whether the marker is circular or not.

Number of markers
The Qualisys camera needs to store the segments in a temporary memory when

trying to find the markers that are non-circular. This memory can be full if there are too many markers that are on the same horizontal level in the 2D view and then the camera will stop looking for non-circular markers. There is no certain number of markers that always work so you need to try it in your setup, but usually less than 10 markers can be processed in the camera. In the case that the camera cannot process all marker there is a red warning in the left upper corner of the 2D view and all the markers that the camera has not been able to process are considered to be OK.

Camera capture rate
A faster camera capture rate decreases the available time to process the markers in the camera, so that not all markers can be processed. In that case there is a red warning in the left upper corner of the 2D view and all the markers that the camera has not been able to process are considered to be OK.

Twin systems

The Twin system feature enables QTM on one computer to control a camera system that is connected to another computer. With this feature you can capture data from two systems and then process them in the same QTM file. It is for example useful if you have an underwater and above water system or if you want to capture a small volume within the larger volume at a higher frequency.

For information about the settings for twin systems see chapters “Twin System” on page 189 and “Twin System Calibration” on page 191.

How to use frame synchronized twin systems with separate volumes

Follow this procedure to use frame synchronized twin systems with separate volumes, for example a system above water and a system underwater.

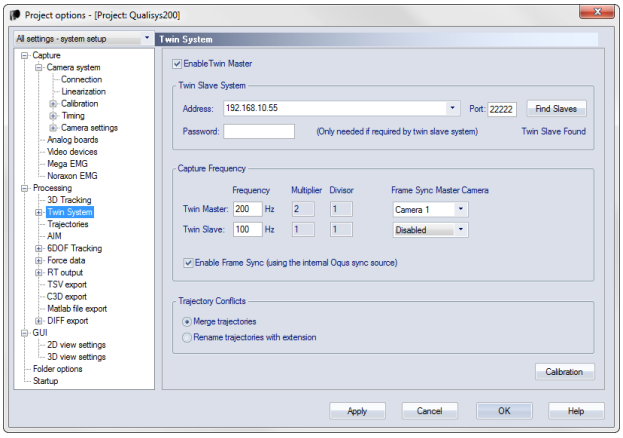
Requirements

1. Two Qualisys systems (Qualisys cameras of any type or mixed).
   a. For Oqus systems at least one camera must be from the 3+-, 4-, 5+-, 6+ or 7+-series if you want to use frame synchronization.
2. Synchronization devices or cables. Each system needs its own Camera Sync Unit or Oqus control splitter.
3. A trigger button with BNC T-splitter and BNC cable.
4. Two computers with two Ethernet ports each.

Procedure

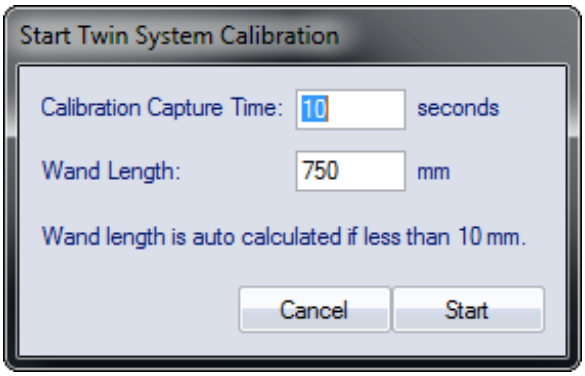
1. Set up the two camera systems and connect them to two separate computers.
   a. It is important that there is a large plane that connects the two volumes so that you can move the wand in the Twin calibration.
2. Calibrate the two systems separately.
3. Connect the free Ethernet ports of the two computers so that they can communicate with each other. Use for example one of the alternatives below:
   a. Connect them both to the same internal network.
   b. Set static IP addresses on the Ethernet connections and connect a cable directly between the computers.
4. Select the system that are going to use the highest frequency as Twin Master system. Usually that is the above water system in above/underwater twin system.
5. Connect the Synchronization output (**Out 1** on Camera Sync Unit or **Sync out** on control splitter) of the device that is selected as **Frame Sync Master Source** to the Synchronization input (**SYNC** on Camera Sync Unit or **Sync in** on control splitter) of the other system.
   Note: You can use the Camera Sync Unit or an Oqus 3+-, 4-, 5+- or 7+-series camera in the Twin slave system as **Frame Sync Master Source**.
6. Connect the same trigger button to **Trig in** (control splitter) or **Trig NO** input (Camera Sync Unit) on both of the systems.

7. Open the **Project options** dialog for the Twin Master system.
   a. Go to the **Twin system** page. For more information about the settings, see chapter “Twin System” on page 189.



   b. Select the option **Enable Twin Master**.
      i. Click on **Find Slaves** and select the computer from the list that appears in the **Address** option.
   c. Select **Enable Frame Sync**
      i. Enter the desired frequencies of the **Twin Master** and **Twin Slave** systems.
      ii. Select the device that is the **Frame Sync Master Source**. It should be is the device generating the **Sync output**. Use the upper drop-down box for the devices in the Twin master system and the lower box for the devices in the Twin slave system.
   d. Select **Merge trajectories**.
   e. Click **OK** to close **Project options**.
8. The systems are now connected. You do not need to change anything on the Twin slave system, all of necessary settings are controlled by the Twin master system.
9. Start the measurement with **Ctrl+N** on the Twin master system. Both systems will start up in preview. The first time you start a measurement you will have to wait while the two systems synchronize to the 100 Hz signal.
10. The systems then need to be twin calibrated to align the coordinate systems. For more information on the twin calibration, see chapter “Performing a Twin calibration” on page 307.

a. Click on **Calibrate Twin System** on the **Capture** menu.



b. Enter the **Calibration capture time** you want to use for the calibration. It must be long enough to move the wand twice in the whole plane between the two systems.

c. Enter the **Wand length** that is used for the twin calibration. If you enter the 0 the wand length is auto calculated but then you must make sure that both markers are in one of the camera systems at the beginning of the measurement.

d. Click **OK**. The twin calibration is done at the frequency of the Twin slave system, therefore the Twin master system might need to synchronize to the 100 Hz signal again.

e. Start the twin calibration with the trigger button when you are ready.

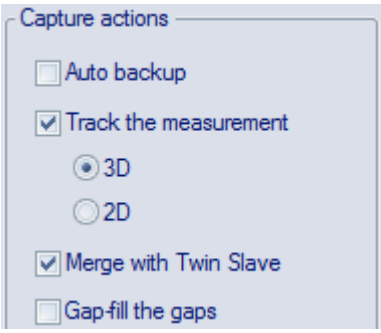
f. Then move the wand with one marker in each camera system. It is extremely important to move the wand so that you cover as much as possible of the plane between the two systems.

g. When finished you will get a result where it is important to check that the wand length is close to what you expect.

11. Activate the processing steps that you want to use.

a. Make sure to always activate 3D tracking on both systems. On the Twin slave system it is actually not necessary to activate anything else.

b. **Merge with Twin Slave** must be activated on the Twin master. Otherwise no data is transferred automatically.



c. Gap-fill, AIM and 6DOF and AIM is best to only activate on the Twin master. It can be done on the Twin slave, but it can result in unnecessary dialogs that interrupts the merging process.

d. The export is of course best to do from the Twin master system.

12. You can now start doing measurements controlled by the Twin master system.
    a. The saving settings on the **Start capture** dialog controls the behavior of both the Twin master and Twin slave system. This means that it is recommended to save the measurements automatically, because then the Twin slave file is merged automatically with the Twin master file. The Twin slave file is then saved both on the Twin master computer and the Twin slave computer.
    b. The Twin master file will contain the 3D data from both systems and can be processed just as any QTM file. The data from the Twin slave system will be labeled with the type **Measured slave** in the **Trajectory info** windows. For information on how to work with the twin files see chapter “Working with QTM twin files” on page 309.
    c. Any video on the Twin slave system is not transferred to the Twin master computer. Video files recorded with the slave system can be added to the merged capture later by importing the video link via the menu **File > Import > Link to Video File**.

       Note: Video recordings on the slave system will only be saved on the slave computer if automatic saving of the capture is enabled on the master system.

       Note: If you import a video from a Qualisys camera that is included in the calibration of the slave system, the calibration information will not be transferred. This means that you cannot apply 3D overlay to the imported video.

How to use start synchronized twin systems with separate volumes

Follow this procedure to use start synchronized twin systems with separate volumes, for example a system above water and a system underwater. Only use this option if you use Oqus systems without any 3+-, 4-, 5+-, 6-+ or 7+-series camera, because the data will be more accurate if the systems are frame synchronized, see chapter “How to use frame synchronized twin systems with separate volumes” on page 301.

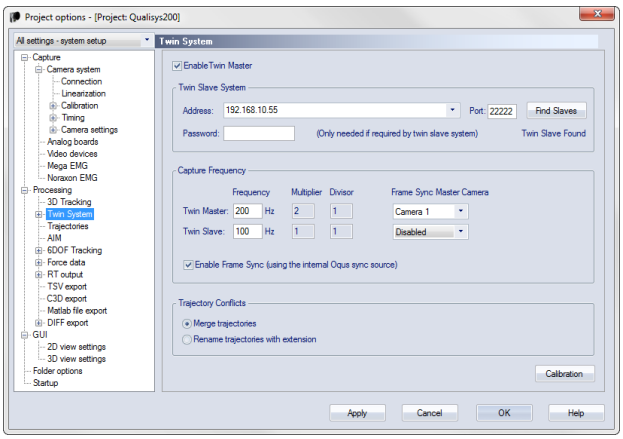
Requirements

1. Two Qualisys systems (Qualisys cameras of any type or mixed).
2. Synchronization devices or control splitters.
    a. Master system: Camera Sync Unit or Oqus control splitter.
    b. Slave system: Camera Sync Unit or Oqus control splitter.
3. A trigger button with BNC T-splitter and BNC cable.
4. Two computers with two Ethernet ports each.

Procedure

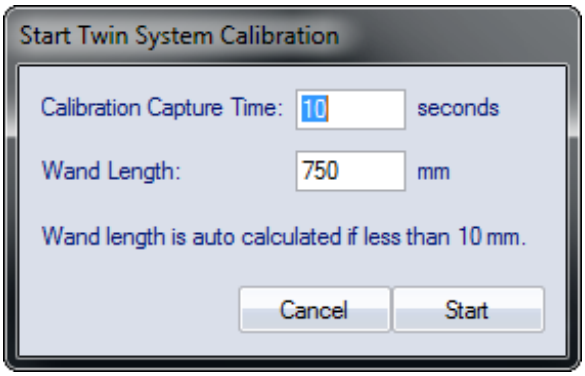
1. Set up the two camera systems and connect them to two separate computers.
    a. It is important that there is a large plane that connects the two volumes so that you can move the wand in the Twin calibration.
2. Calibrate the two systems separately.

3. Connect the free Ethernet ports of the two computers so that they can communicate with each other. Use for example one of the alternatives below:
   a. Connect them both to the same internal network.
   b. Set static IP addresses on the Ethernet connections and connect a cable directly between the computers.
4. Select the system that are going to use the highest frequency as Twin Master system. Usually that is the above water system in above/underwater twin system.
5. Connect the same trigger button to the **Trig NO** input (Camera Sync Unit) or **Trig in** (Oqus control splitter) on both of the systems.
6. Open the **Project options** dialog for the Twin Master system.
   a. Go to the **Twin system** page. For more information about the settings, see chapter "Twin System" on page 189.



   b. Select the option **Enable Twin Master**.
      i. Click on **Find Slaves** and select the computer from the list that appears in the **Address** option.
   c. Enter the desired frequencies of the **Twin Master** and **Twin Slave** systems. Do not select **Enable Frame Sync**.
   d. Select **Merge trajectories**.
   e. Click **OK** to close **Project options**.
7. The systems are now connected. You do not need to change anything on the Twin slave system, all of necessary settings are controlled by the Twin master system.
8. Start the measurement with ctrl+n on the Twin master system. Both systems will start up in preview.
9. The systems then needs to be twin calibrated to align the coordinate systems. For more information on the twin calibration, see chapter "Performing a Twin calibration" on page 307.

a. Click on **Calibrate Twin System** on the **Capture** menu.



b. Enter the **Calibration capture time** you want to use for the calibration. It must be long enough to move the wand twice in the whole plane between the two systems.

c. Enter the **Wand length** that is used for the twin calibration. If you enter the 0 the wand length is auto calculated but then you must make sure that both markers are in one of the camera systems at the beginning of the measurement.

d. Click **OK**.

e. Start the twin calibration with the trigger button when you are ready.

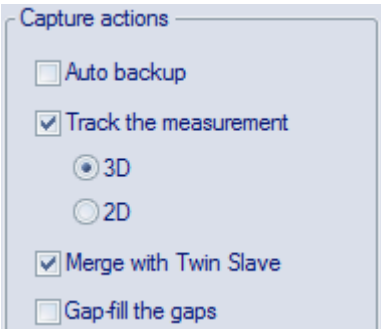
f. Then move the wand with one marker in each camera system. It is extremely important to move the wand so that you cover as much as possible of the plane between the two systems.

g. When finished you will get a result where it is important to check that the wand length is close to what you expect.

10. Activate the processing steps that you want to use.

a. Make sure to always activate 3D tracking on both systems. On the Twin slave system it is actually not necessary to activate anything else.

b. **Merge with Twin Slave** must be activated on the Twin master. Otherwise no data is transferred automatically.



c. Gap-fill, AIM and 6DOF and AIM is best to only activate on the Twin master. It can be done on the Twin slave, but it can result in unnecessary dialogs that interrupts the merging process.

d. The export is of course best to do from the Twin master system.

e. Any video on the Twin slave system is not transferred to the Twin master computer.

11. You can now start doing measurements controlled by the Twin master system.
    a. The saving settings on the **Start capture** dialog controls the behavior of both the Twin master and Twin slave system. This means that it is recommended to save the measurements automatically, because then the Twin slave file is merged automatically with the Twin master file. The Twin slave file is then saved both on the Twin master computer and the Twin slave computer.
    b. The Twin master file will contain the 3D data from both systems and can be processed just as any QTM file. The data from the Twin slave system will be labeled with the type **Measured slave** in the **Trajectory info** windows. For information on how to work with the twin files see chapter “Working with QTM twin files” on page 309.

Performing a Twin calibration

When using Twin systems it is necessary to specify the relations between the coordinate systems of the Twin master and Twin slave systems. Therefore the two systems must first be calibrated separately with wand calibration. The twin calibration is then created to transform the Twin slave data to the coordinate system of the Twin master. The relations are usually established in a Twin calibration procedure. However it can also be entered manually if actual relations are not critical.

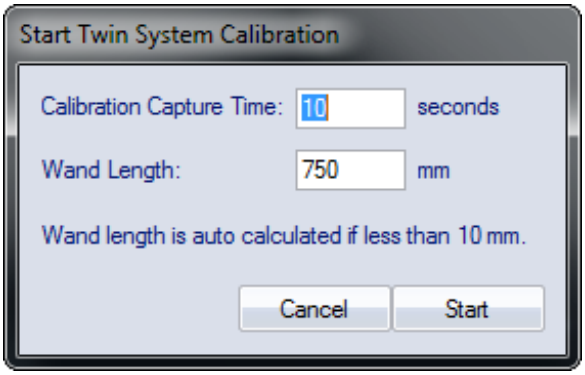
Note: If the two systems share any volume it is recommended to place the L-frame for the calibration there. Then you do not need to transform the Twin slave data at all, since the two systems share the same coordinate system definition.

To perform a Twin calibration you need to first make sure that you have made the following preparations.

1. Set up the two systems so that the Twin master controls the Twin slave, see chapter “How to use frame synchronized twin systems with separate volumes” on page 301. And make sure that both systems are calibrated.
2. Make sure that you have a wand that reaches between the two volumes. It is recommended to use one where you know the length, but you can also use one of the systems to measure the wand during the twin calibration.
3. Make sure that the plane between the two systems are as large as possible. Since a larger plane will give a more accurate twin calibration.

Follow this procedure for the twin calibration:

1. Start a measurement with **Ctrl+N** on the Twin master system.
2. Click on **Calibrate Twin System**on the **Capture** menu.



3. Enter the **Wand length** and the number of seconds needed to at least move the wand in the whole plane twice.

   Note: If you don't have a wand length you can enter 0 and then make sure that both markers are clearly visible in one of the systems when you start the twin calibration.

4. Click on **Start** and wait for the systems to synchronize. The twin calibration is always done at Twin slave frequency, so the Twin master system might need to synchronize before you can start.

5. Press the trigger button to start the twin calibration and start moving the wand.

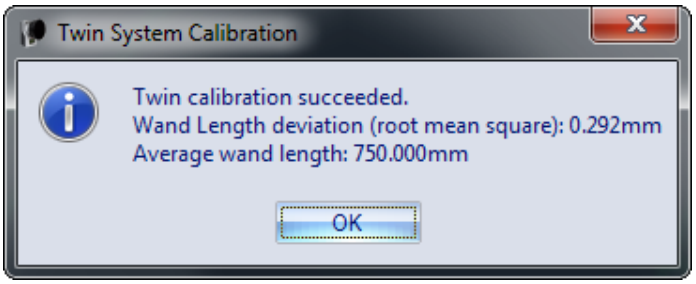
6. Move the wand so that you have one marker in each volume. The twin calibration will be better if you fulfill these requirements:

   a. Try to move it so that you cover the whole plane between the two volumes.

   b. Move the wand at least twice in the whole plane.

   c. Make sure that the angle of the wand differs when you move it around in the volumes.

7. When finished the Twin slave data is transferred automatically and then you get the result of the calibration. Make sure that the wand length is close to what you have entered and that the standard deviation is not much higher than the deviations of the regular calibrations.



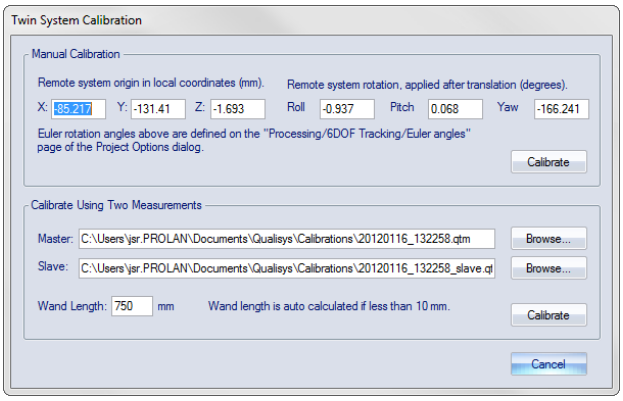
   a. The twin calibration fails if the wand length differs more than 1 % of the entered value. The reason could be that other markers alters the twin calibration. Open the files that are saved in the calibration folder of the project and check the 3D data. Delete any 3D data that is not the wand markers. Then reprocess the Twin calibration according to the instructions below.

8. If you want to check the actual translation and rotation of the slave data , open the **Twin system** page on the Twin master system and then click on **Calibration**.

It is possible to reprocess the Twin calibration from the **Project options** dialog or in the reprocessing dialogs. Follow these steps to change the twin calibration:

1. Click on **Calibration** on the **Twin system** page to open the **Twin System Calibration** page.

2. Then click on **Calibrate** to open the **Twin system calibration** dialog, see chapter “Twin System Calibration dialog” on page 192.



3. In this dialog you can change the QTM files and wand length used for the Twin calibration.
   a. The twin calibration files are saved in the Calibration folder of the current project. Open the file and retrack it if you think that errors in the tracking caused problems with the twin calibration. After you retracked the file you need to reprocess the twin calibration.
   b. You cannot reprocess the twin calibration if you do not have the files created in the twin calibration. However if you make sure to save all of your data in the same project then they are always saved in the Calibration folder of that project.
4. Then click on **Calibrate** inside the **Calibrate Using Two Measurements** heading to reprocess the twin calibration.
   Note: You can also enter a new translation and rotation manually under the **Manual Calibration** heading, but it is not recommended if you want the best accuracy.

Working with QTM twin files

The complete data from a QTM twin system measurement is always stored in two separate files. If you have used the **Merge with Twin slave** option in the processing, the slave file automatically gets the same name as the master with the ending _slave. When the twin slave file is merged with the twin master file, it is then only the 3D data of twin slave that is merged into the file. The twin slave cameras are also displayed as greyed cameras in the 3D view.

The merged twin slave 3D data is marked as **Measured slave** in the trajectory info windows, but that is the only difference compared with the other data. You can then identify and work with all of the 3D data as usual, for example apply an AIM model to all the data. However if you need to retrack the data you need to do that separately in the twin slave file, since the 2D data is not imported to the master file.

Since only the 3D data is merged from the Twin slave the following processing and input must be performed on the twin master.

Video (Miqus, Oqus or external video devices)

Analog data (Analog board or EMG)

Calculate 6DOF

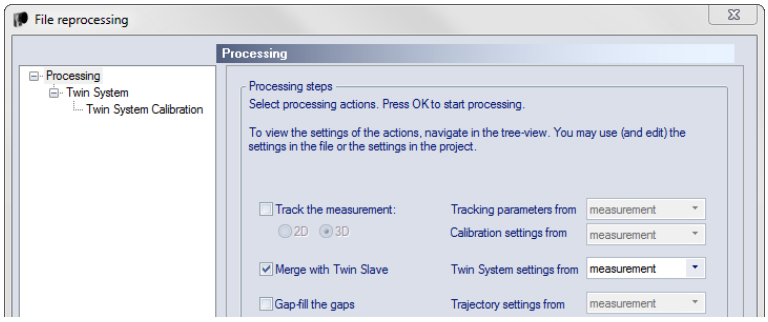
Calculate force

Export data

Merging Twin files in reprocessing

The data of two QTM files can always be merged in reprocessing if you have not been using the Merge with Twin slave option or if you need to redo it because the tracking of the Twin slave file has changed.

Note: All of the previous Twin slave data is deleted when you merge twin slave data in reprocessing.



The merging is always done by processing the Twin master file and activating the **Merge with Twin slave** option on the **Processing steps** list. You can edit both which Twin slave file that is used and the Twin calibration before you reprocess the file. The twin calibration that is used in the reprocessing is shown on the **Twin system calibration** page. You can change the twin calibration by clicking the **Calibrate...** button, but you must make sure to click the correct **Calibrate** button (Manual or Using two measurements) in the **Twin system calibration** dialog to apply the change, for more information on the Twin calibration settings see chapter “Twin System Calibration” on page 191.

If you use the **Reprocess** option on the **Capture** menu to merge the files, then you copy the **Twin slave file** location either from **measurement** or **project**. However you can edit the file location and the Twin calibration on the **Twin system** page.

If you instead use the **Batch process** option on the **File** menu, which file is used is directly controlled by where you copy the settings from.

processed file
The Twin slave file and all other Twin system settings are copied from the file that is processed and cannot be changed.

project and **present file**
The processing automatically uses the file with the same name and the _slave extension. However you can edit the Twin calibration if you need to.

Twin 3D data interpolation

It is recommended to use the same frequency in the two systems to avoid any interpolation. If you have to use a lower frequency for the slave system then it is recommended that you use a divisor of the twin master frequency to minimize the interpolation.

The interpolation is a linear interpolation meaning that a line is drawn between the twin slave 3D data. Then any time where the twin slave data does not exist the linear data will be used instead. The interpolated data will have a zero residual so that you can distinguish it in post processing.

Twin system with a shared volume

It is possible to use a twin system with a shared volume, if you want to capture at a higher frequency in parts of the volume. For example if you want to capture the impact of a golf swing.

In this case it is best to place the L-frame in the shared volume so that you can easily merge the data of the two systems. You can then reset the calibration on the **Twin system calibration** dialog. There can still be a difference in the data of the two systems that is less than a mm. If you find a constant difference you can change the twin calibration data manually to improve the match. It is not possible to use the automatic twin calibration.

When merging the data in a shared volume case, it is often best to identify the data in both of the files before merging them in reprocessing, see chapter “Working with QTM twin files” on page 309. Then if you use the **Merge trajectories** option the twin slave data will be used automatically where the twin master data is missing. Any labeled twin slave data that overlap some twin master data will be deleted and if you want to use it you have to remove the twin master data that is wrong and merge the files again.

Real time mode

Real time in QTM

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, OSC or with VRPN.

At the moment there are Qualisys made interfaces for mobile devices (Android, iOS), Motion Monitor, Motionbuilder, LabView and Matlab. Please contact Qualisys AB if you want to use these interfaces. There are also externally made plugins for Visual3D, EON Reality and Vizard. Please contact the company for these programs if you want to use one of these plug-ins.

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.

Open source software, including several SDKs (e.g. C++, C#, Python) and connect modules (e.g. for Unity, Unreal), are available at the Qualisys site on Github, see https://github.com/qualisys/. For detailed information, see the documentation included in the respective packages.

The real time performance depends on the computer and graphical card specifications, the number of cameras, the number of markers and the complexity of the used AIM model(s), amongst others. For more information about how it works, see chapter "How real time works in QTM" below.

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)

Note: There are buffers in the analog equipment which means that there will not always be new analog data on every new marker frame.

Force data (Force, Moment and COP)

Note: There are buffers in the analog equipment which means that there will not always be new force data on every new marker frame.

6DOF data (Position, Rotation matrix, Euler angles, Residuals)

Skeleton data (T-pose, Segment positions and orientations in global or local coordinates)

Parameters (e.g. Label name, Statistics for the RT performance, Force plate settings)

Events

Images (from cameras in Marker intensity and Video mode; only supported in preview mode, not when streaming data from file)

QTM control(QTM can also be controlled via the RT protocol, e.g. to start/stop measurements and to change camera mode.)

OSC data see chapter “OSC integration” on page 319.

VRPN see chapter “VRPN integration” on page 320.

These are the different steps in the real-time process, for some advice on how to setup the real time see chapter “Outline of how to use real time” on the facing page.

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”.

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”.

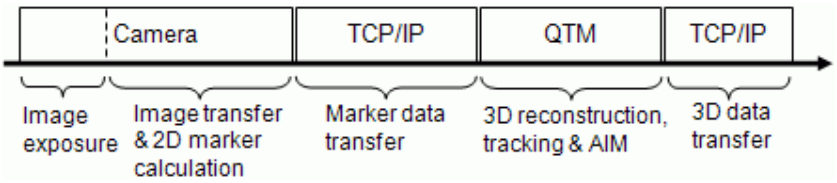
Use the following guidelines to get as fast real-time as possible:

- 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 latency

Real time latency is the time it takes from the marker exposure to the time 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.

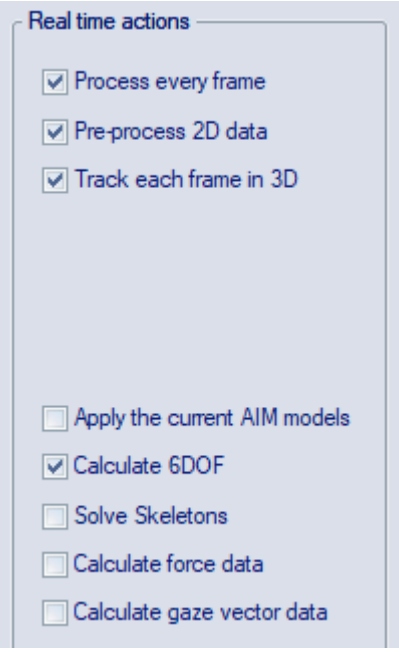
Because the latency is system and computer dependent it is not possible to give a latency for any system. However for Oqus cameras there is a very elegant way of monitoring the system latency integrated in QTM. Enable the display of latency with the **Show Realtime Latency** option on the **GUI** page 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.

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.

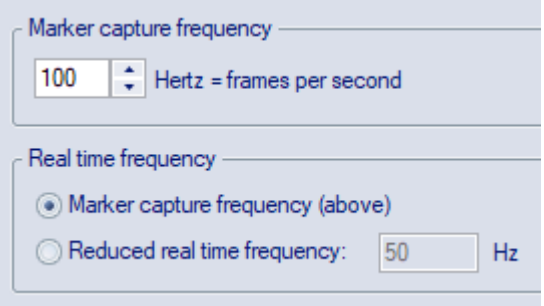
1. Before you start the real-time, make sure that you have an AIM model or 6DOF bodies for the movement that you are going to capture, see chapter "Automatic Identification of Markers (AIM)" on page 364 and "6DOF tracking of rigid bodies" on page 336, respectively.
   Note: It is best if the AIM model 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 example, for Visual3D typically the following actions are used: **Pre-process 2D data**,

Track each frame: 3D, **Apply the current AIM models** and **Calculate force data**.



Note: When the option **Process every frame** is unchecked, QTM will skip frames in case the real time processing exceeds the frame duration.

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 mainly on the computer specifications, as well as the complexity of the AIM model.



Note: Using a too low real-time frequency may lead to suboptimal 3D tracking depending on the velocity of the movements.

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 on the **GUI** page in the **Projects Options** dialog. This will reduce the processing capacity needed by QTM.

c. AIM and 6DOF data can be restarted with the **F9** shortcut or 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 or client, see the following chapters for examples.

Server mode

QTM can be started in server mode to minimize the interference by QTM when running an RT client. To start QTM in server mode use the command-line argument **/server** when starting QTM. The modified behavior for the server mode is as follows:

- QTM is started minimized.
- Non-critical dialogs are hidden.
- Crash dumps are automatically sent to Qualisys without user feedback.

Mobile apps

There are two mobile apps available for Android and iOS devices, which can connect to QTM. The apps can be downloaded from Google Play or the Apple App Store.

Viewfinder
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 a Qualisys camera.

QTM Remote
The QTM 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.

To communicate with QTM the mobile device must be connected via WiFi to the same local network as the QTM computer. Check with your system administrator if you need help with this.

Note that for controlling QTM the app must be connected as a master to QTM, see chapter “Master/Slave mode in mobile apps” below

Master/Slave mode in mobile apps

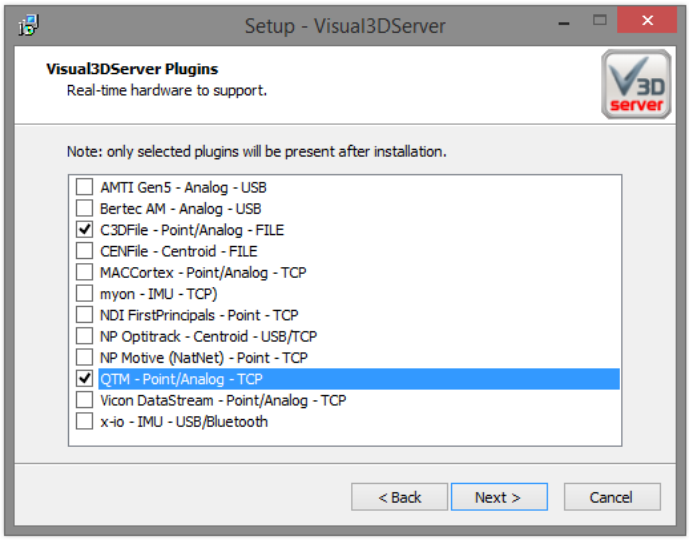
To use all of the features in the Viewfinder and the Remote app, you must be connected as master to QTM. Only one mobile device or other program that uses the RT output at a time can be master. When the mobile app is master it can change settings and control measurements in QTM. If the mobile app is 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 “Real-Time output” on page 227. 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.

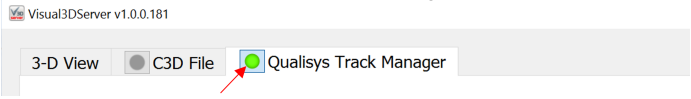
Visual3D integration

The following steps describe how to stream real-time data from QTM to Visual3D. It is recommended to run Visual3D on another computer connected to the same network as the QTM computer to optimize the performance.

1. If QTM and Visual3D run on separate computers, you need to know the IP address of the QTM computer. On the computer running 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 **Details**. Write down the IP address (IPv4 Address).
2. QTM must be running in real-time (preview) mode, and markers must be identified automatically by AIM.
3. In Visual3D follow these steps:
   a. For the first use, install Visual3D RT Server from your C-Motion client login. If it is not available, please contact support@c-motion.com. During installation, select the QTM plugin.

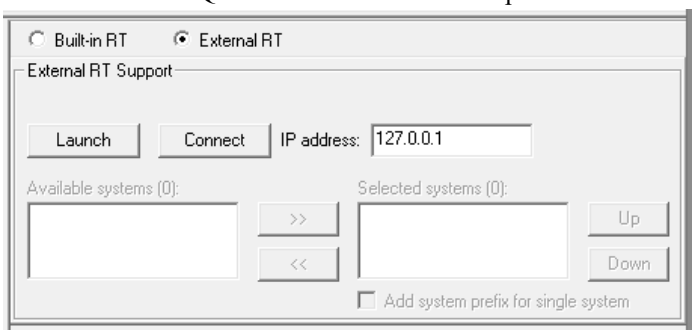


   b. Start Visual3D Server and click the circle in the header of the Qualisys Track Manager tab to connect to QTM.

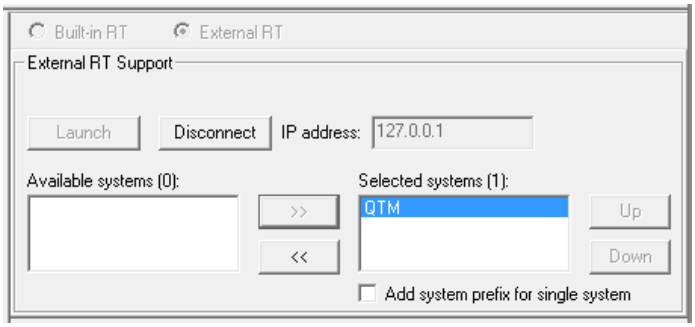


   c. Start Visual3D and switch to the Real-time tab.

d. Select **External RT** and fill in the IP address of the QTM computer. Use 127.0.0.1 in case QTM runs on the same computer.



e. Click **Connect** and use the >> button to move QTM from *Available Systems* to *Selected Systems*.



A tutorial for how to add processing in Visual3D can be found here (http://c-motion.com/v3dwiki/index.php?title=Tutorial_RT_Biofeedback_-_Qualisys_QTM). More information about the settings in Visual3D Server can be found here (http://c-motion.com/v3dwiki/index.php?title=RT_Plugin_Qualisys_QTM).

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 check the QTM RT protocol.pdf and a demo program for Max included in the RT protocol folder in the Qualisys Track Manager folder.

To record synchronized sound with the motion capture it is recommended to use the equipment described in chapter “Using Oqus sync unit for synchronization” on page 486.

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.

LabVIEW, Matlab and Simulink integrations

The LabVIEW, Matlab and Simulink integrations provides an easy integration with the QTM RT server and to access the motion capture data in real time. The Matlab interface is an easy way of retrieving and processing the data in real-time. For close-loop control systems or similar, Simulink or Labview is more suited for the task.

All data types in QTM is available, which include unidentified 3D, identified 3D, 6DOF, analog data,force data unlinearized 2D and linearized 2D. The LabVIEW plugin

can also send some commands to QTM, for example to start and stop measurements. All of the plug-ins contain example applications of how you can access the data from QTM.

The plugins are designed for control purposes and is therefore optimized for speed and low latency. Because of this, data is transmitted over UDP, an unreliable data transmission protocol, meaning that packets may be lost. This is ok for the purposes of a control process since there is no meaning to receive old data. However, it also means that the plugins are not the best method to save data from QTM.

The plugins require a license key in QTM, please contact sales@qualisys.com for a quote.

VRPN integration

VRPN is a device-independent and network-transparent system mostly used in VR application, for example in EON reality and Worldviz's Vizard. QTM supports VRPN output of 6DOF data and their markers in RT. For more information about the protocol check the VRPN information on vrgeeks.com and http://www.cs.unc.edu/Research/vrpn/index.html. There is also an example script for Vizard in the section “Vizard example script” below.

The 6DOF data can be accessed via the VRPN protocol during preview and during a measurement, for more information about how to use real time in QTM see chapter “How real time works in QTM” on page 313.

Note: The coordinate system in the VRPN data is always a right handed system with the y-axis pointing up. This means that how the coordinate system is orientated in QTM does not affect the VRPN data. For example Worldviz's Vizard uses a left handed coordinate system and the data must therefore be converted, see example script.

Note: Both EON Reality and Vizard have made thier own plugins that uses the standard QTM RT protocol. Contact EON Reality respectively Worldviz for more information.

Vizard example script

The following is an example python script for Vizard. Note that it will only work with the two rigid bodies called Body_1 and Body_2 in QTM.

```
import viz
import vizact
import vizshape
viz.go()

#Define position of the camera. The Y axis is pointing upwards.
viz.MainView.setPosition([1,10,0])
viz.MainView.lookAt([0,0,0])

#Define boxes for the 6DOF bodies
box1 = vizshape.addBox([0.1,0.1,0.05],splitFaces=True,pos=(0,0.5,4))
box2 = vizshape.addBox([0.1,0.1,0.05],splitFaces=True,pos=(0,0.5,4))
```

```
box1.color(1,0,0)
box2.color(0,1,0)

#Define spheres for the individual markers
b1_1 = vizshape.addSphere(0.05)
b1_2 = vizshape.addSphere(0.05)
b1_3 = vizshape.addSphere(0.05)
b1_4 = vizshape.addSphere(0.05)
b1_1.color(100,0,0);
b1_2.color(0,100,0);
b1_3.color(0,0,100);
b1_4.color(100,100,100);

b2_1 = vizshape.addSphere(0.1)
b2_2 = vizshape.addSphere(0.1)
b2_3 = vizshape.addSphere(0.1)
b2_4 = vizshape.addSphere(0.1)
b2_1.color(100,0,0);
b2_2.color(0,100,0);
b2_3.color(0,0,100);
b2_4.color(100,100,100);

#Define the input from QTM. Change localhost to the ip address of the QTM computer
if it is not the same as the Vizard computer.
vrpn = viz.add('vrpn7.dle')
trk1 = vrpn.addTracker('Body_1@localhost') #Default sensor is 0
trk1_1 = vrpn.addTracker('Body_1@localhost', 1)
trk1_2 = vrpn.addTracker('Body_1@localhost', 2)
trk1_3 = vrpn.addTracker('Body_1@localhost', 3)
trk1_4 = vrpn.addTracker('Body_1@localhost', 4)
trk2 = vrpn.addTracker('Body_2@localhost') #Default sensor is 0
trk2_1 = vrpn.addTracker('Body_2@localhost', 1)
trk2_2 = vrpn.addTracker('Body_2@localhost', 2)
trk2_3 = vrpn.addTracker('Body_2@localhost', 3)
trk2_4 = vrpn.addTracker('Body_2@localhost', 4)

#Add global axes
vizshape.addAxes()
```

```
#Add axes on the boxes for the 6DOF bodies.
vizshape.addAxes(parent=box1, scale=[0.3,0.3,0.3])
vizshape.addAxes(parent=box2, scale=[0.3,0.3,0.3])

#Convert the right hand input to the left hand system in Vizard. You need to use both
swapPos and swapQuat.
trk1.swapPos([1,2,-3])
trk1.swapQuat([1,2,-3,-4])
trk1_1.swapPos([1,2,-3])
trk1_1.swapQuat([1,2,-3,-4])
trk1_2.swapPos([1,2,-3])
trk1_2.swapQuat([1,2,-3,-4])
trk1_3.swapPos([1,2,-3])
trk1_3.swapQuat([1,2,-3,-4])
trk1_4.swapPos([1,2,-3])
trk1_4.swapQuat([1,2,-3,-4])

trk2.swapPos([1,2,-3])
trk2.swapQuat([1,2,-3,-4])
trk2_1.swapPos([1,2,-3])
trk2_1.swapQuat([1,2,-3,-4])
trk2_2.swapPos([1,2,-3])
trk2_2.swapQuat([1,2,-3,-4])
trk2_3.swapPos([1,2,-3])
trk2_3.swapQuat([1,2,-3,-4])
trk2_4.swapPos([1,2,-3])
trk2_4.swapQuat([1,2,-3,-4])

#Define what to display in the Vizard 3D GUI.
def setSensorData():
    box1.setPosition(trk1.getPosition())
    box1.setQuat(trk1.getQuat())
    b1_1.setPosition(trk1_1.getPosition())
    b1_1.setQuat(trk1_1.getQuat())
    b1_2.setPosition(trk1_2.getPosition())
    b1_2.setQuat(trk1_2.getQuat())
    b1_3.setPosition(trk1_3.getPosition())
    b1_3.setQuat(trk1_3.getQuat())
    b1_4.setPosition(trk1_4.getPosition())
```

```
        b1_4.setQuat(trk1_4.getQuat())

        box2.setPosition(trk2.getPosition())
        box2.setQuat(trk2.getQuat())
        b2_1.setPosition(trk2_1.getPosition())
        b2_1.setQuat(trk2_1.getQuat())
        b2_2.setPosition(trk2_2.getPosition())
        b2_2.setQuat(trk2_2.getQuat())
        b2_3.setPosition(trk2_3.getPosition())
        b2_3.setQuat(trk2_3.getQuat())
        b2_4.setPosition(trk2_4.getPosition())
        b2_4.setQuat(trk2_4.getQuat())

def printSensorData
        print(trk1_1.getPosition())
        print(trk1_2.getPosition())
        print(trk1_3.getPosition())
        print(trk1_4.getPosition())

        print(trk2_1.getPosition())
        print(trk2_2.getPosition())
        print(trk2_3.getPosition())
        print(trk2_4.getPosition())

vizact.ontimer(0.01,setSensorData)
vizact.ontimer(1,printSensorData)
```

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 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 project 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, see chapter “Reprocessing a file” on the facing page, or automatically on saved files by using**Batch processing**, see chapter “Batch processing” on page 329.

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 and batch process.

The following processing steps are available on the page:

1. **Auto backup**, see chapter “Auto backup” on page 284.
2. **Pre-process 2D data**, see chapter “Processing 2D data” on page 330.
3. **Track the measurement**, see chapter “Tracking the measurements” on page 334.
   a. **3D**, see chapter “3D tracking measurements” on page 334.
   b. **2D**, see chapter “2D tracking of data” on page 360.
4. **Merge with Twin slave**, see chapter .“Working with QTM twin files” on page 309.
5. **Gap-fill the gaps**, see chapter “Trajectories” on page 192.
6. **Apply the current AIM model**, see chapter “Automatic Identification of Markers (AIM)” on page 364.
7. **Calculate 6DOF**, see chapter “6DOF tracking of rigid bodies” on page 336.
8. **Solve Skeletons**, see chapter “Tracking of skeletons” on page 350.
9. **Calculate force data**, see chapter “Force data calculation” on page 386.
10. **Calculate gaze vector data**
11. **Export to TSV**, see chapter “Export to TSV format” on page 389.
12. **Export to C3D format**, see chapter “Export to C3D format” on page 397.
13. **Export to Matlab file**, see chapter “Export to MAT format” on page 399.
14. **Export to AVI**, see chapter “Export to AVI file” on page 404.
15. **Export to FBX file**, see chapter “Export to FBX file” on page 406.

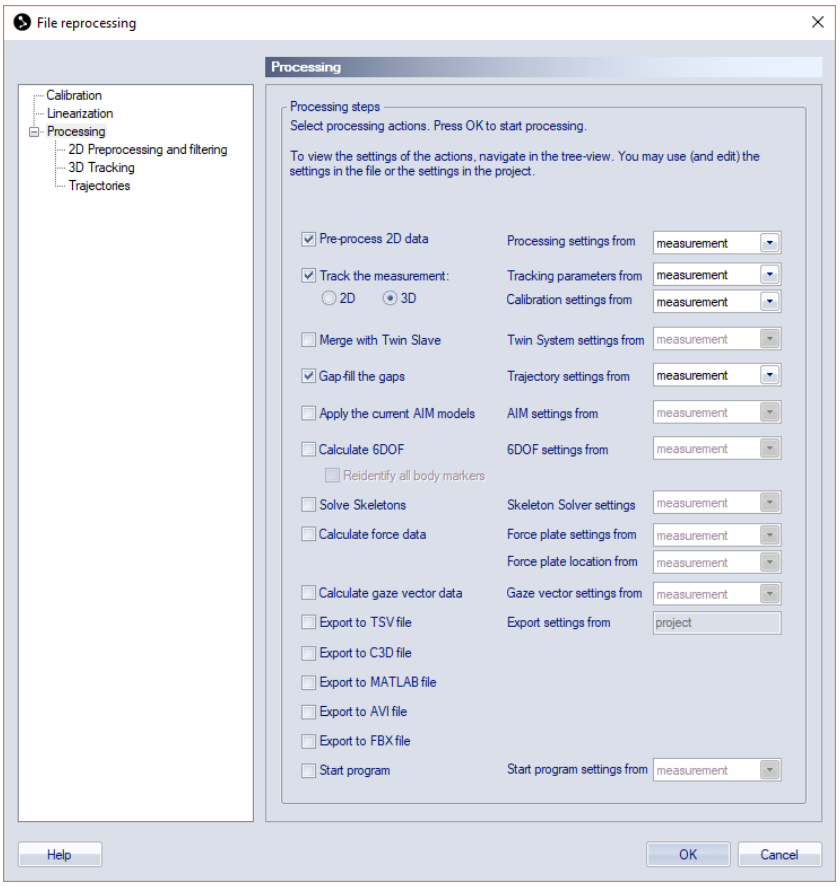
The settings of each processing step can be found in the tree to the left in the dialog. For information about the settings see each respective chapter under “Project options” on page 121.

Important: The two tracking options under **Track the measurement** are mutually exclusive and can therefore not be used at the same time.

Reprocessing a file

If there are problems with the tracking of a measurement , it can be reprocessed with for example other **Tracking**, **Calibration** and **Linearization** parameters than during the capture. Retracking 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 181. 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 project, 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 project and can be edited in the tree-view to the left. Editing the settings will change the current project settings as well.

Important: When **Track the measurement** with **3D** or **2D** is used all of the trajectories will be unidentified again and all manual editing of the trajectories will be lost.

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 **project**.

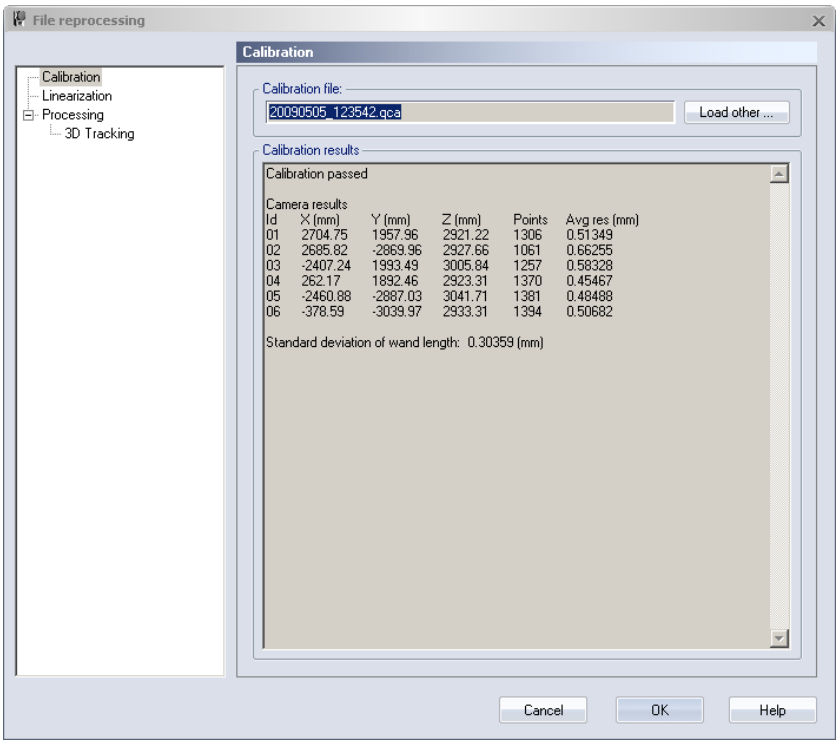
Note: The **Calculate force data** settings are split into **Force plate settings** and **Force plate location** so that the source can be set separately.

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 141.

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 279. 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.

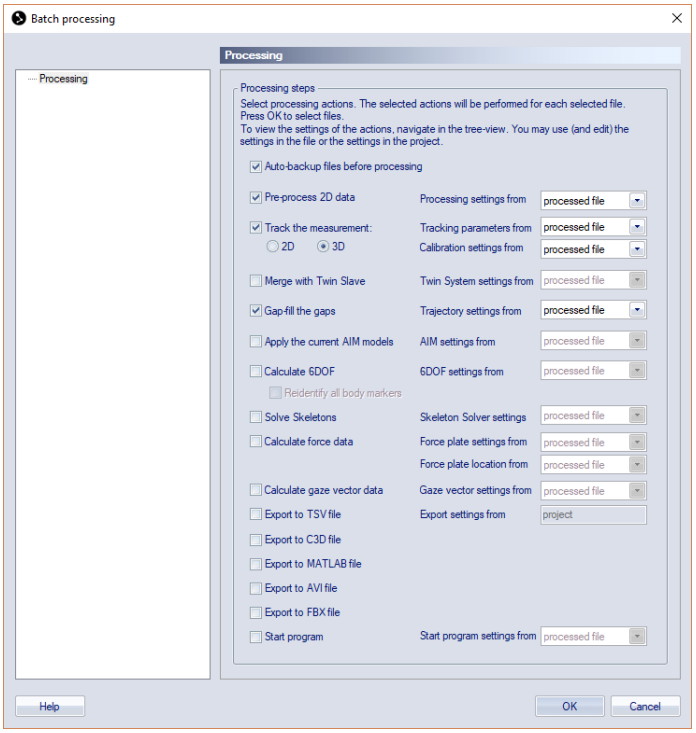
Batch processing

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 and all manual editing of the trajectories will be lost.

To start a batch process, follow the steps below.

1. Click **Batch process** on the **File** menu. Select the files and click **Open** 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 181.

   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 project, 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 project settings to each processed file. The settings can be edited in the tree-view to the left. Editing the settings will change the current project settings as well. This option is often the best to use when you want to change the settings of the processed files.

Note: If **Merge with Twin slave** is used with **Project** settings then the twin master file is automatically merged with a file with the same name and ending with _slave that is located in the same folder.

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** and **Calculate force data**.

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 **project**.

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 **project** 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**. 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.

Processing 2D data

The 2D data is processed when the **Pre-process 2D data** option is enabled on the **Processing** page in **Project Options**. When reprocessing 2D data on a file then the 3D, 6DOF and Skeleton data must also be reprocessed for the changes to have an affect.

There are three processing options for 2D data:

- Circularity filter
- Software marker masks
- Marker size filter

How to use circularity filter

The circularity filter can be used to correct or delete bad or partially hidden markers. It is only available when the **Marker circularity filtering** option is enabled for the camera, see chapter “How to use non-circular marker settings” on page 299.

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 the non-circular markers. This will result in more 3D data, which otherwise cannot be calculated.

The filter has two options: **Correct center point of non-circular markers** and **Discard non-circular markers.**. One of the options must be selected for the 2D preprocessing. Use the following advice to set the settings for the different options.

Correct center point of non-circular markers
Any markers that have been filtered out by the camera as non-circular will be fitted to a circle if possible and merged markers are also corrected.

Circularity level
In this mode you can set the **Circularity level**, in the camera settings, depending on how many markers you have. Try to find a level where the camera has time to process the markers.

Effect on 3D data
This mode will give you more exact 3D data than the unfiltered data, because the corrected markers have more exact center points than the camera data. It has most effect if only two cameras can see the marker. For example you can use this mode to improve the accuracy of the calibration, since the wand markers are partially hidden by the stick.
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.

Discard 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, in the camera settings, according to how much you want to delete.

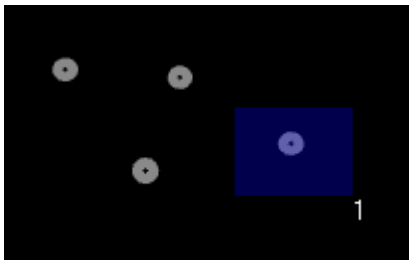
Effect on 3D data
This mode will give you less 2D data than the unfiltered data and actually most of the time it is better to try to correct the markers. The biggest advantage with this mode is that it is faster than correcting the markers. So if you have many cameras that can see each marker you will not loose much 3D data and it will also be more exact.

Note: If you change this option when processing a file, then you must also reprocess the 3D, 6DOF and skeleton data for the change to have any effect.

How to use software marker masks

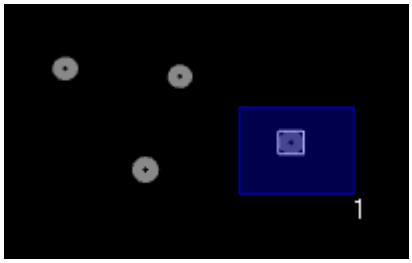
The software marker masks can be used to remove markers in an area of the 2D data. It works best for noise that static is in the same area throughout the measurement.

Add software marker masks to a file with the **Marker mask** tool in the 2D view. The mask is added as a blue square. Use the **Selection** tool in the 2D view to resize and move the mask with the pointer. There can be up to 100 masks per camera. Software marker masks are drawn linearized, this means that, unlike camera marker mask, it is best to keep the **Show linearized data** option enabled on the **2D view settings** page.

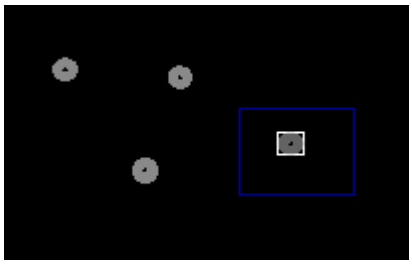


To activate the mask the QTM file must be reprocessed with the **Pre-process 2D data** and **Software marker mask** options enabled. After processing, the mask is surrounded by a blue square to show that the mask is applied. The markers that are filtered are surrounded by a white square. The 3D, 6DOF and skeleton data must also be reprocessed for the change to have any effect.

Note: Reprocessing a file with **Software marker mask** disabled will keep the masks, but they are not applied and the markers are not filtered. The masks will look like when they are added without the blue square.

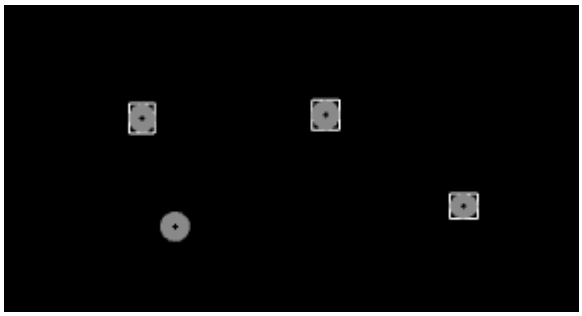


Right-click on a mask and select **Delete this mask** or **Delete all masks** to delete a mask respectively all masks for that camera. When an applied mask is deleted, the marker is still filtered (shown by the white square) and the edge around the mask is shown. The file needs to be reprocessed with **Software marker mask** enabled for the 2D data to be restored.



How to use marker size filter

The 2D data can be filtered on the size of the markers. Set the minimum and/or maximum size in subpixels for the markers in the file with the options: **Minimum marker size** and **Maximum marker size**. When a marker is removed by the filter it is displayed with a white square in the 2D view and it is not displayed in the 2D data in the Data info window.



To remove the size filtering reprocess the file with **Pre-process 2D data** enabled and the options**Minimum marker size** and **Maximum marker size** disabled.

Note: If the marker size filter is changed when processing a file, then the 3D, 6DOF and skeleton data must also be reprocessed for the change to have any effect.

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.

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 184.

How to set the 3D tracker parameters depends a lot on the measurement setup, for some advice see chapter “Advice on how to set 3D Tracker parameters” below. For an example on how to test that the tracking is working properly, see chapter “3D tracking test” on the facing 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 327. For information about how to correct errors in the tracking process see chapter “Troubleshooting tracking” on page 497.

Advice on how to set 3D Tracker parameters

The default parameters for the 3D tracker will lead to good tracking results in most situations. However, in some situations, when you notice that the tracking is not optimal, changing the 3D tracker parameters may help to improve the results. When changing these parameters, it is beneficial to have a good understanding of how they influence the tracking. For more information about this, see chapter “3D Tracker parameters” on page 184.

Here follows some general advice on how the 3D tracking parameters can be adapted for special tracking situations.

Prediction error
The **Prediction error** is by default set to 25 mm. If you have problems with switching of trajectories it may help to use a lower value. For very fast movements, for example tracking a golf club during a swing, you can use a higher value.

Note: When capturing very fast movements it is important to use a high capture rate for optimal 3D tracking results.

Maximum residual
The **Maximum residual** is by default set to 6 mm. For very large tracking volumes it may help to increase the value for more robust tracking. Reducing the value may help to avoid switching artifacts, but may lead to more fragmented trajectories. A good way to estimate the **Maximum 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.

Minimum trajectory length
The minimum trajectory length is by default set to 2 frames, which is the minimum value for a part or trajectory. Increasing the value may help to reduce the amount of ghost markers, for example when tracking highly reflective objects.

Minimum ray count per marker
The minimum ray count is by default set to 2, which means that a marker can be defined in 3D when seen by two cameras. Increasing the value to 3 or more cameras may help to reduce the amount of ghost markers, for example when tracking highly reflective objects. However, this may come with the disadvantage of more fragmented trajectories.

Ray length limits
The ray length limits are by default calculated automatically based on the calibration, but they can also be set manually. Using a certain minimum ray length can help to avoid ghost markers close to the cameras if there are many spurious reflections, for example from a water surface. For very large tracking volumes, it can be useful to set a maximum ray length so that cameras that are at a large distance of a marker are excluded from the calculation of its position.

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 269.
- Check camera positioning, incorrect positioning results in hidden markers and problems with the calculation of markers; see chapter “Camera positioning” on page 254.
  Note: The system must be calibrated after the rearrangement.
- Check the tracking parameters; see chapter “3D Tracker parameters” on page 184.
  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 262.
  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 of rigid bodies

6DOF tracking identifies rigid bodies from the 3D data of a measurement and calculates their translation and rotation. 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 196.

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 *pose*, namely 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 the next page. 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 three points, which should not lie on the same line. When designing a 6DOF body, consider the following aspects.

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)

Choose an appropriate marker size for the used measurement volume. It is recommended that the markers as seen by the cameras (**xSize** and **ySize** in the **Data info** window) are at least 256 subpixels.

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

The following requirement must be fulfilled by the 6DOF body for the 6DOF tracking to work.

- 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.

- 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.
- 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.

- Put the markers sufficiently high so that they are not obscured by the model when it rolls away from the camera.
- 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.

- 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 200.
- 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 or moving rigid object. This option is recommended for creating new rigid bodies.

a. Select the trajectories associated with the rigid body in the **3D view window** or the **Trajectory info window**.

b. Right-click on the selection and choose **Define rigid body (6DOF)** from the context menu. Alternatively, use the keyboard shortcuts **F8** or **Shift + F8**. There are two options:

   Current Frame (Shift + F8): Create rigid body definition based on marker configuration of the current frame.

   Average of frames (F8): Create rigid body definition based on the average configuration of the markers in the current capture. The advantage of this method is that the statistics of the marker positions are taken into account. The **Bone tolerance** setting of the body will be based on these statistics.

c. Specify the name of the rigid body.

The new rigid body definition is added to the **Rigid bodies** lists in the **Project options** dialog and to the current file. The default origin is the geometric center of the points included in the rigid body definition. The default orientation is set to that of the global coordinate system at the beginning of the capture. Optionally, you can modify the local coordinate by translation and rotation (see chapters “Translate body” on page 199 and “Rotate body” on page 200) or change the reference coordinate system (see chapter “Coordinate system for rigid body data” on page 202).

Note: When defining a new body in this way, the 6DOF data is re-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 196.
4. Acquire the points through a capture in preview mode, see chapter “Acquire body” on page 203. This method is described in chapter “Define 6DOF bodies using Acquire body method” 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.

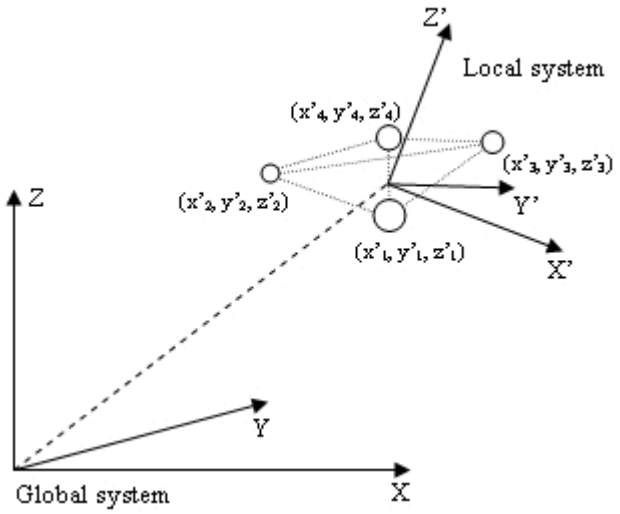
1. The bodies must use exactly the same markers. with the same names, in its respective definition.
2. The bodies may however have different names.

Define 6DOF bodies using Acquire body method

Follow these steps to acquire a new 6DOF body in preview mode:

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 rotation 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 200 and “Translate body” on page 199.
   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 202.
    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. The local coordinate system is by default placed in the geometric center of the points.

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 196.

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 202 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

Rigid bodies can be tracked in real time and in a capture. To track rigid bodies, **3D tracking** and **Calculate 6DOF** must be enabled on the **Processing** page in the **Project options** dialog. You can also use reprocessing to edit the 6DOF data, see chapter “Calculating 6DOF data” on the next page.

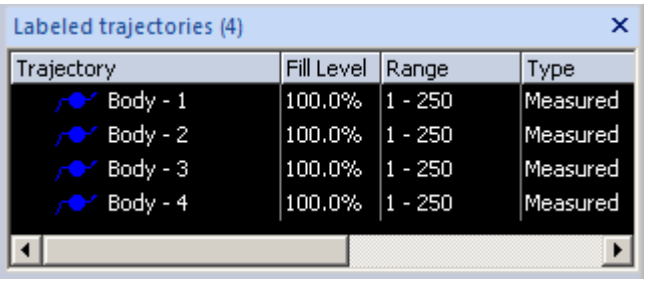
The rigid body tracking behavior is defined by the settings in the **6DOF Tracking** page in the **Project options**. The main parameters for controlling the tracking are **Bone tolerance** and **Max. residual**. These parameters specify the tolerance for deviations of measured trajectories from the rigid body definition. Optionally, 6DOF data can be smoothed by choosing one of the available **Filter** presets, see chapter “Smoothing 6DOF data” on page 204. The filtering is applied both in real time and in a capture.

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 according to the

definition of the 6DOF body, 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 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 6DOF tracker will automatically move trajectories that are outside the **Bone tolerance** to the unidentified labels. This is done in RT as well on tracking a file.
    - If you have identified trajectories manually in a file the 6DOF calculation will try to use all of the data, and then there will be no 6DOF data in frames where a marker is outside the tolerance. To make the 6DOF calculation remove the data outside the tolerance you can activate the **Reidentify all body markers** option while reprocessing, see below.
- If there are frames 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.
- When a **Filter** preset has been selected, the 6DOF data will be smoothed according to the preset parameters both in real time and in a capture, see chapter "Smoothing 6DOF data" on page 204.

The 6DOF data can be recalculated 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.

Note: To delete all 6DOF data in a file you can reprocess the file with **Calculate 6DOF** with an empty list of rigid bodies.

When reprocessing a file the 6DOF data is reprocessed in the following ways depending on the processing steps.

Calculate 6DOF without the **Reidentify all body markers** option

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.

Calculate 6DOF with the **Reidentify all body markers** option

All cases
With the reidentify 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, then those trajectories will keep their identity.

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 196.
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 206. 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:

  $-180° \leq \text{Roll} \leq 180°$

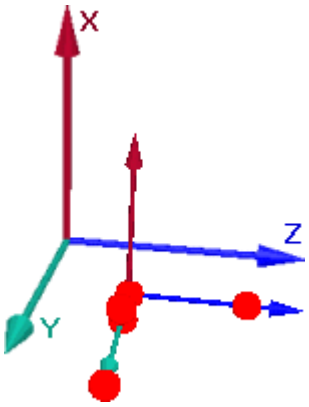
  $-90° \leq \text{Pitch} \leq 90°$

  $-180° \leq \text{Yaw} \leq 180°$

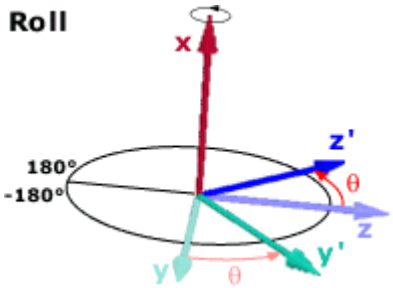
  In these ranges, roll, pitch and yaw are unambiguous and can describe any orientations of a rigid body.

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 "Appendix F: Rotation angle calculations in QTM".

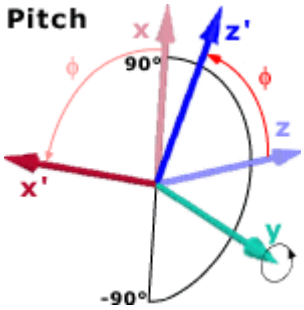
Below follows an example to show the definitions of the rotation angles. It starts with a 6DOF body, which is in alignment with the global coordinate system.



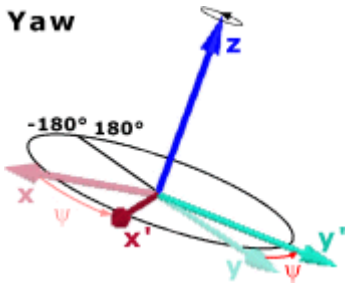
First the local coordinate system is rotated around the X-axis (roll) with an angle θ to the new positions y' and z' of the Y- and Z-axis.



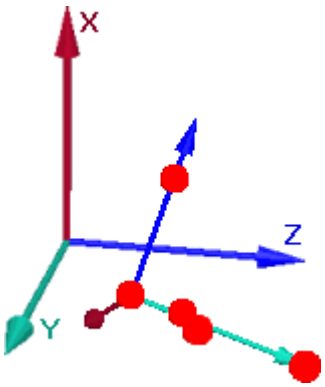
After the roll the local coordinate system rotates around the Y-axis (pitch) with the Y-axis in its new position. The X- and Z-axis is rotated with an angle ϕ to the new positions x' and z'.



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 F - 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 313.

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.

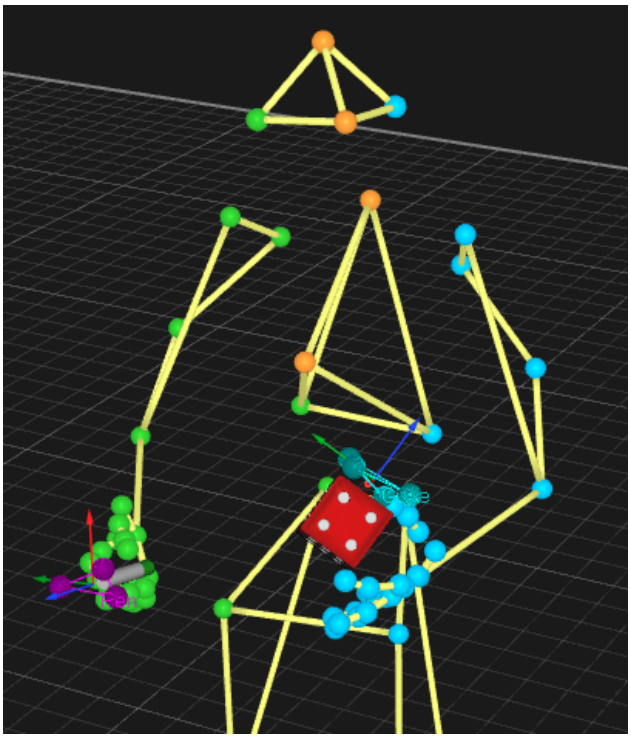
The data values that will be used are selected on the **6DOF Analog export** page, see chapter “6DOF analog export” on page 228. Since the required board, see chapter “Analog output hardware” below, has 16 channels the output is limited to 16 data values of 6DOF bodies. In order to maximize the use of 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.

Rigid body meshes



A rigid body mesh can be used to visualize the object tracked by a rigid body. QTM supports meshes in the format of the Wavefront 3D object (.obj) files. To use a 3D mesh copy the .obj file and all of the related .mtl and texture files to the *Meshes* folder of the project. The mesh can be associated with a rigid body by the following two ways.

From 3D view window
Right-click in the 3D view and select **Rigid Body -> Change Mesh of Rigid Body**.

From Project Options
Double click on the **Mesh** column fro the rigid body on the **6DOF tracking** page.

The **Rigid Body Mesh Settings** dialog is opened, see chapter "Rigid body Mesh Settings dialog" on page 205. Select the mesh file to use from the list and modify the settings. Use the **Apply** button to verify that it looks correct in the 3D view. It is often best to use trial and error to find the position and rotation for the mesh. Use these tips to position the mesh.

1. The mesh is always scaled so that the longest side is 1 m, because there is no standard for how to save the size in the .obj file. Use the **Scale** option to set the correct scale for a mesh.
2. Always scale the mesh first before translation and rotation, because the distance to the mesh coordinate system is changed when scaling.

Note: Using really large meshes can slow down the rendering in QTM.

Note: The *Meshes* folder can be set to any folder on the **Folder options** page in **Project options**. This means that the same meshes can be used by different projects.

Sharing a file with a rigid body mesh

When sharing a file with a rigid body mesh it is important to include the mesh files with the QTM file. This can be done in the following ways:

- Share the whole project.
- Share the QTM file and mesh files together. QTM will find the mesh files if they are located in the same folder as the QTM file. The other user can also copy the mesh files to the *Meshes* folder in their project.

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 337 and “Define 6DOF bodies using Acquire body method” on page 339.
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 341.
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 208.
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 (Oqus only)” on page 350.
- A special case for 6DOF bodies is if you need multiple sets of 6DOF data, i.e. multiple 6DOF definitions, 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 364. 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 364. 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 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 are 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 have not moved, 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 three markers that do 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 (Oqus only)

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 non-circular marker settings” on page 299.

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.

Tracking of skeletons

The following chapters describe how to define and track skeletons in QTM. The two main applications of skeleton tracking are animation, and sports biomechanics. The specific workflow may depend on the application. Generally, tracking of skeletons involves the following steps.

Choice of marker set
Skeleton tracking requires the use of a dedicated marker set. For more information about the available marker sets and the possibilities to customize them, see chapter “Marker sets for skeleton tracking” on the facing page.

Setting up the AIM model for automatic labeling of markers
Automatic labeling of markers is an important part of an efficient workflow. The optimal approach may depend on the application. For the recommended approaches, see chapter “Automatic labeling of markers for skeleton tracking” on page 354.

Skeleton calibration
Create a skeleton definition or update an existing one, see chapter “Skeleton calibration” on page 356.

Modifying the skeleton definition
Edit the skeleton definition to change the scaling factor, marker weights or segment definitions, see chapter “How to modify the skeleton definition” on page 358.

Measuring skeleton data
Real time tracking and capturing of skeleton data, see chapter “How to measure skeleton data” on page 360.

Exporting and streaming
Export or stream skeleton data for further processing in an external program or real time applications, see chapter “Export and streaming of skeleton data” on page 360

Marker sets for skeleton tracking

Skeleton tracking in QTM requires the use of a designated marker set. There are two marker sets available. The choice depends mainly on the application.

Animation
The recommended marker set for animation is the Qualisys Animation Marker Set, see chapter “Qualisys Animation Marker Set” below.

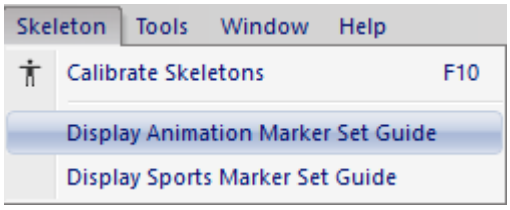
Sports and biomechanics
The recommended marker set for sports and biomechanics applications is the Qualisys Sports Marker Set, see chapter “Qualisys Sports Marker Set” on the next page.

The marker sets can be customized in the following ways.

- You can add extra markers, for example for more robust tracking of specific movements. For more information, see chapter “Adding extra markers to a skeleton” on page 353.
- You can change the names of the default labels, see chapter “Skeleton marker label mapping” on page 353.

Qualisys Animation Marker Set

The *Qualisys Animation Marker Set* is dedicated to character animation applications using the Qualisys Skeleton Solver. For detailed information about the Qualisys Animation Marker Set, you can open the marker set guide via the **Skeleton** menu.



The marker set guide contains information about:

- The markers and their placement.
- The use of optional markers for a better definition of the orientation of the skeleton segments.
- T-pose requirements, see also chapter “T-pose” on page 357.

- The use of extra markers, see also chapter “Adding extra markers to a skeleton” on the facing page.

Two AIM models for the Qualisys Animation Marker Set are included in the installation of QTM in the subfolder *Models\AIM*:

Animation.qam
Generic AIM model with the default markers.

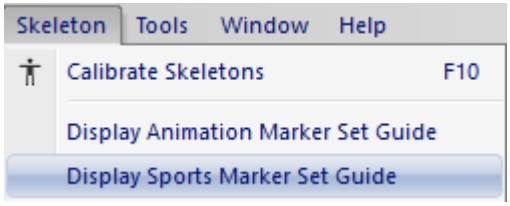
Animation_Optional.qam
Generic AIM model including both default and all the optional markers as described in the *Animation Marker Set Guide*.

For more information about automatic labeling of the markers, see chapter “Automatic labeling of markers for skeleton tracking” on page 354.

It is possible to use alternative labels for the markers. This can be useful if you have existing files with a different markers set. For more information on how to use a different mapping of markers, see chapter “Skeleton marker label mapping” on the facing page.

Qualisys Sports Marker Set

The *Qualisys Sports Marker Set* is dedicated to sports and biomechanics applications using the Qualisys Skeleton Solver. As opposed to the animation marker set, the segments associated with the sports marker set are defined in a conventional and biomechanical way which helps to compute and interpret the joint angles more easily. For detailed information about the Qualisys Sports Marker Set, you can open the marker set guide via the Skeleton menu.



The marker set guide contains information about:

- The markers and their placement.
- The use of static markers required for the skeleton calibration.
- The use of optional extra markers, see also chapter “Adding extra markers to a skeleton” on the facing page.

Two AIM models for the Qualisys Sports Marker Set are included in the installation of QTM in the subfolder *Models\AIM*:

Sports_Static.qam
Generic AIM model including static markers for the skeleton calibration.

Sports_Dynamic.qam
Generic AIM model without static markers for dynamic measurements.

Both AIM models are pre-trained for trouble free automatic labeling of a wide range of movements and actors. For guidelines on how to use these AIM models in different scenarios, see chapter “Using AIM for sports and biomechanics” on page 355.

It is possible to use alternative labels for the markers. This can be useful if you have existing files with a different markers set. For more information on how to use a different mapping of markers, see chapter “Skeleton marker label mapping” on the facing page.

Adding extra markers to a skeleton

Extra markers can be placed on the subject and included in the skeleton definition. This option can be useful for improving the tracking of the segments of the skeleton or for creating unique marker configurations when capturing multiple actors. Follow these steps to add extra markers to a skeleton:

- The label must have the same prefix as the skeleton, e.g., *JD_...* for a skeleton with name *JD*. The prefix should end with an underscore; the first underscore in the label will be considered as the separator for the skeleton name.
- The marker will be automatically associated to a segment based on its placement. However, if you want to specify a specific segment to which the marker will be associated you can include the segment name in the marker label, e.g., *JD_Hips_...* for a marker associated with the *Hips* segment of the skeleton definition of *JD*.

  The respective segment names are: *Hips*, *RightUpLeg*, *LeftUpLeg*, *RightLeg*, *LeftLeg*, *RightFoot*, *LeftFoot*, *RightToeBase*, *LeftToeBase*, *Spine*, *Spine1*, *Spine2*, *Neck*, *Head*, *RightShoulder*, *LeftShoulder*, *RightArm*, *LeftArm*, *RightForeArm*, *LeftForeArm*, *RightHand*, *LeftHand*.

  Note: When using the *Qualisys Sports Marker Set*, the segments *RightToeBase*, *LeftToeBase*, *Spine1*, *Spine2* and *Neck* are locked, so make sure that extra markers are not assigned to these segments if you want these markers to contribute to the tracking of the skeleton. You can use the segments *RightFoot*, *LeftFoot*, *Spine* and *Head* instead.
- After calibrating the skeleton, QTM displays a list of the assignment of segments to the extra markers, which you can use to check if the assignments are correct.

The use of extra markers will depend on your camera configuration and the type of movements you want to capture. For the Animation marker set, here are some recommendations for the body part location of extra markers:

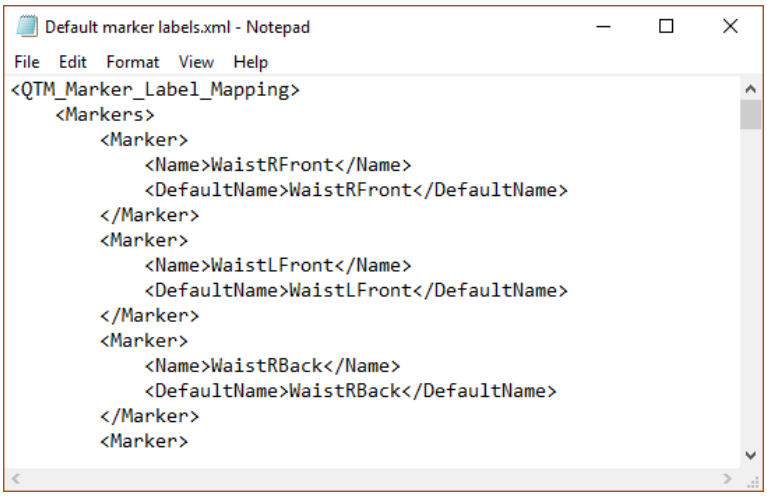
- On *RightHand* and *LeftHand*
- On *Spine2* (above *Chest* marker)
- On *RightShoulder* and *LeftShoulder* (clavicles)
- On *Spine1* (below *BackL* and *BackR* markers)

Skeleton marker label mapping

The skeleton solver automatically recognizes the default marker labels of the *Qualisys Animation Marker Set* and the *Qualisys Sports Marker Set*. If you want to use alternative marker labels, for example to apply the skeleton solver to existing measurements, you can create a custom mapping. If you are using an alternative marker set, the marker positions should correspond closely to those described in the marker guides of the Qualisys marker sets for Animation and Sports.

Follow these steps to create a custom mapping.

1. Go to the **Skeleton Solver** page in the **Project Options**.
2. Save the default skeleton marker label mapping by pressing the **Save** button. The file contains all labels of both the Animation and Sports marker sets.
3. Open the created file in a text editor (for example Notepad).



4. Edit the marker names contained in the <Name> tags and save the file when done.
5. Load the file in the **Skeleton Solver** page under the **Marker Label Mapping** heading by pressing the drop down menu and choose **Custom (Load from file)**.

Automatic labeling of markers for skeleton tracking

To facilitate automatic labeling of markers, generic AIM models for the supported marker sets are included in QTM. The AIM models are stored in the QTM installation (default location: *C:\Program files (x86)\Qualisys\Qualisys Track Manager*) in the subfolder *Models\AIM*. For information about how to use AIM for different applications, refer to the following chapters:

For animation applications with the Qualisys Animation Marker Set, see chapter “Generating an AIM model for animation” below.

For sports and biomechanics applications with the Qualisys Sports Marker Set, see chapter “Using AIM for sports and biomechanics” on the facing page.

To apply one or more AIM model(s) follow these steps:

- Add the model(s) to the **Applied models** list on the **AIM page** in **Project Options** (see chapter “AIM” on page 193 for more information).
- Make sure that the *Apply current AIM models* is checked as an action (real time and capture) in the **Processing** page under **Project Options**.

Generating an AIM model for animation

This chapter describes the best practice for using AIM for skeleton tracking for animation.

It is highly recommended to create a specific AIM model for each actor to be tracked. To create a new AIM model, follow these steps:

1. On the **AIM page** in **Project Options**, add one of the generic AIM models for animation to **Applied models**. The following generic AIM models are available in the QTM installation in the subfolder *Models\AIM*:
   a. The file *Animation.qam* contains an generic AIM model for the default markers of the Qualisys Animation Marker Set.

   b. The file *Animation_Optional.qam* contains a generic AIM model for the default markers and all the optional markers described in the *Animation Marker Set Guide*.

2. Make a capture of the actor. A good practice is to make a capture including a *T-pose* and a *Range of Motion* sequence, which can be used afterward to calibrate and validate the skeleton definition.
3. Make sure that the labeling is correct.
4. If you are using extra markers, add the labels manually to include them in the new AIM model.
5. Add a prefix to labels. You must end the prefix with an underscore, e.g., *JD_* for a skeleton with name *JD*.

   Note: The first underscore in a trajectory label will be used as the separator for the skeleton name.
6. Create a new AIM model, see chapter "Generating an AIM model" on page 364 for more detailed information.

If needed, you can improve the AIM model by training it with more movement data, see chapter "Generating an AIM model" on page 364 for more detailed information.

If you want capture multiple actors, repeat the above steps for each actor. When done, add all actors to be included in the capture to **Applied models** on the **AIM page** in **Project Options**.

Using AIM for sports and biomechanics

There are two generic AIM models for the sports marker set available in the installation of QTM in the subfolder *Models\AIM*:

- The file *Sports_Static.qam* contains all markers, including the static markers that must be included for the skeleton calibration
- The file *Sports_Dynamic.qam* contains all markers that are needed for skeleton tracking.

To apply one of these AIM models, add it to the **Applied models** list on the **AIM page** under **Project Options**.

The best way to use AIM for sports and biomechanics depends on your specific application. Two possible use scenarios are suggested below, but of course you can choose an alternative approach that works best for your application.

Scenario 1
Use the generic AIM models included with QTM for calibration and tracking. The generic AIM models include a standard prefix *Q_* for all the marker labels, corresponding to a standard skeleton name *Q*. This scenario is only suitable when measuring a single person.

For the static pose:

Use the static AIM model *Sports_Static.qam* for labeling the static pose. The static AIM model includes the static markers that are required for the skeleton calibration.

For the dynamic measurements there are the following options:

- Use the static AIM model *Sports_Static.qam* if you want to keep the static markers during the dynamic measurements.

- Use the dynamic AIM model *Sports_Dynamic.qam* if you want to do the dynamic measurements without the static markers.

Note: It is not possible to automatically label extra markers with the generic AIM models. You need to create a new AIM model if you want to include extra markers.

Note: The prefix of the marker labels can be changed by editing the AIM file (.qam) in a text editor using search and replace.

Scenario 2
Create a specific AIM model for an individual person. This scenario is similar to the one described for animation applications, see chapter “Generating an AIM model for animation” on page 354.

For the static pose:

1. Use the generic static AIM model *Sports_Static.qam* for automatic labeling of the first capture for skeleton calibration.
2. Optionally, you can rename the prefixes of the markers for creating a skeleton definition with a different name. The easiest way to do this is to select the trajectories in the trajectory info window, remove the prefix of the selected trajectories and add a new prefix. Do not forget to use an underscore character "_" as a separator.
3. Use a static pose to calibrate the skeleton, see chapter “Skeleton calibration” below.

For the dynamic measurements:

1. Capture a file with a range of motion. Use one of the generic AIM models to label it, with or without static markers, depending on the marker setup you want to use for your dynamic trials.
2. Rename the prefixes so that they correspond to the skeleton name.
3. If you want to use extra markers you need to manually label them. Use the same prefix if you want to include them in the skeleton definition.
4. Create a new AIM model, see chapter “Generating an AIM model” on page 364 for more detailed information.

If needed, you can improve the AIM model by training it with more movement data, see chapter “Generating an AIM model” on page 364 for more detailed information.

If you want capture multiple persons, repeat the above steps for each person. When done, add all persons to be included in the capture to **Applied models** on the **AIM page** in **Project Options**.

Skeleton calibration

Skeleton calibration needs to be done to fit a skeleton definition to the measured marker positions. The skeleton calibration can be applied in real time during preview or to a recorded capture.

Before applying the skeleton calibration, make sure that the actor stands in the correct calibration pose. The pose depends on marker set.

- For the *Qualisys Animation Marker Set* a **T-pose** is required for a correct calibration. Optional markers can be added for a better definition of the orientation of the segments. For more information, refer to the marker set

guide that can be opened in QTM via the **Skeleton** menu.

- For the *Qualisys Sports Marker Set* make sure that the static markers are present. The actor or subject should stand in upright position with both feet flat on the floor. A specific pose is not required, since the marker positions in the Qualisys Sports Marker Set provide sufficient information for correct definition of the skeleton segments.

Press the **Calibrate skeletons** button (keyboard shortcut **F10**) to apply the skeleton calibration. When applying the skeleton calibration for the first time, a new skeleton definition will be added to the **Skeleton Solver** page under **Project Options**. When the skeleton is already defined, the skeleton calibration will replace the existing skeleton definition in the project.

If there are multiple actors in the preview or capture, the skeleton calibration is applied simultaneously to all actors.

When the skeleton calibration is applied to a capture, the new skeleton definition will be automatically applied to the whole file.

It is possible to calibrate and solve a part of the skeleton, see chapter “Partial skeleton calibration” below.

Once the skeleton is calibrated it is possible to modify the skeleton definition, see chapter “How to modify the skeleton definition” on the next page.

Note: To apply a new or updated skeleton definition to existing captures they need to be reprocessed using the **Skeleton Solver** settings of the project.

T-pose

When using the *Qualisys Animation Marker Set*, the actor must stand in a correct *T-pose* when applying the calibration. When applying the skeleton calibration to a file, make sure that the actor stands in T-pose in the current frame. For a correct T-pose, make sure that the following requirements are fulfilled:

- Thighs and shanks must be vertical making a small gap between both ankles.
- Feet must be parallel to each other and pointing in the front of the subject.
- Head and neck must be aligned with the spine, i.e., the subject must stand with the head straight, facing forward.
- Arms, forearms and hands must be parallel to the floor. Check that the *HandOut* and *WristOut* markers are horizontally aligned.
- Palms of the hands must face the floor.
- Arm and forearm must be aligned. A virtual line should go through the glenohumeral joint (underneath *ShoulderTop* marker), the elbow joint and the wrist joint.

Partial skeleton calibration

It is possible to define and solve only a part of the skeleton. This can be useful, for example if you only need lower or upper body skeleton data.

For calibrating a part of the skeleton, make sure to add the skeleton prefix to the markers that should be included in the partial skeleton definition. The following requirements should be met when defining and solving a partial skeleton:

1. The root segment should be included. This is the *Hips* segment in the standard skeleton model associated with the Animation and the Sports marker

sets.

2. All markers needed for defining and tracking the intermediate segments between the root segment and the end segment(s) should be present. The required markers may depend on the used marker set.

It may not always be obvious which markers are required for a partial calibration. For example, for the *Animation* marker set, the *SpineTop* marker is required for the T-pose calibration of the lower body part of the skeleton.

Once the required markers are labeled, calibrating and solving the skeleton works similar as calibrating and tracking the complete skeleton:

1. You can create an AIM model including the markers of the partial skeleton to facilitate tracking, see chapter “Automatic labeling of markers for skeleton tracking” on page 354.
2. When solving the skeleton only the segments included in the skeleton definition will be tracked.
3. You can modify the scale factor or marker weights, see chapter “How to modify the skeleton definition” below.
4. You can export and stream partial skeleton data, see chapter “Export and streaming of skeleton data” on page 360.

An alternative way to apply partial skeleton solving is by removing segments from a complete skeleton definition. For more information, see chapter “How to modify the skeleton definition” below.

How to modify the skeleton definition

Once a skeleton is defined it is possible to influence the tracking by modifying the skeleton definition. The skeleton definition can be modified in two ways:

1. By importing a modified skeleton XML file.
   a. Use the **Save** button in the **Skeleton Solver** page to export the skeleton definition XML file .
   b. Edit the XML file in an external text editor. The format is the same as for the RT protocol, see Skeleton xml parameters.
   c. Use the Load button in the **Skeleton Solver** page to import the modified skeleton definition XML file .
2. By sending an XML packet via the real time protocol, see the **QTM RT Protocol** documentation included in the QTM installation for more information.

Here are some examples of elements that can be modified:

- Scale factor, see chapter “Scale factor” on the facing page.
- Marker weights, see chapter “Skeleton marker weights” on the facing page.
- Marker and segment labels.
- Degrees of freedom of segments.
- Remove segments for partial skeleton solving. Note that the root segment and all intermediate segments between the root and the respective end segments should be present.

Note: To apply a modified skeleton definition to existing captures they need to be reprocessed with the modified skeleton definition.

Note: All modifications will be undone when recalibrating a skeleton in QTM. Recalibration replaces the existing skeleton definition in the QTM project with a new one.

Scale factor

The scale factor can be used to indicate the scale of the skeleton tracked in QTM relative to an animated object, for example, an avatar in an external animation application. The scale factor can be set in two ways:

1. By setting the **Scale factor (%)** value for a skeleton in the Skeletons list on the **Skeleton Solver** page.
2. By modifying the value in the *Scale* tag in the skeleton definition XML and re-importing it in the project. In the XML file the scale factor is set as the ratio.

The scale factor is applied to the following data:

- Real time skeleton data streamed by QTM.
- The **Skeletons** and **Characters** data in the FBX export.

The scale factor is applied inversely. For a scale factor larger than 1 (or 100%) the skeleton data is scaled down, and vice versa, for a scale factor less than 1 (or 100%) the skeleton data is scaled up.

The scale factor is not applied to the skeleton data that is displayed in the **Data info** window.

Skeleton marker weights

It is possible to change the relative weighting of the markers. To do so, follow these steps:

1. Open the **Skeleton Solver** page in the **Project Options**.
2. In the **Skeletons** list, select the skeleton you want to modify.
3. Press the **Save** button to save the skeleton definition at a chosen location. The default file name is the name of the skeleton with extension .xml.
4. Open the skeleton definition file in a text editor.
5. In the Skeleton definition, locate the marker(s) you want to edit and change the value of the *weight* tag. The default value is 1 (in the xml file: <Weight>1.000000</Weight>). It is recommended to change the weight by a factor of maximum 5, i.e., values ranging from 0.2 to 5.
6. Save the file in the text editor.

To apply the modified skeleton definition, you will need to load the skeleton definition.

1. Open the **Skeleton Solver** page in the **Project Options**.
2. Under **Skeletons**, press the **Load** button. In the file dialog, locate the modified skeleton definition file and press **Open**.
3. The modified skeleton definition will now be applied to subsequent measurements of the actor associated with the skeleton definition. To apply the modified skeleton definition to existing captures, you will need to reprocess them with the modified skeleton settings, see chapter “Reprocessing a file” on page 327.

Note: The modifications of the skeleton definition will be undone when recalibrating the skeleton. If you want to apply marker weighting again, you will need to go repeat the above steps for the updated skeleton definition.

How to measure skeleton data

For measuring skeleton data, make sure that the **Solve Skeletons** option is checked under **Processing** in the **Project Options**. Skeleton data can be calculated both in real time and in a capture. The skeleton data can be displayed and plotted in real time and from a file via a **Data info** window, see chapter "Skeleton data information" on page 89 for more information.

Note: The real time skeleton data may differ from the skeleton data in a capture as the process used in real time is optimized for faster calculation.

For animation applications it is good practice to start and end each capture with the actor(s) standing in T-pose. This allows for the possibility to recalibrate the skeleton if the quality of the fit is insufficient, for example if a marker has moved.

During post-processing you can label and edit the trajectories as usual. In addition to the standard gap filling methods, you can apply kinematic gap filling to the markers included in the skeleton definitions, see chapter "Filling of gaps" on page 378. The kinematic gap fill option is available in the **Trajectory Editor window** or via the **Trajectory Info Window menu**.

After you made changes to trajectories, for example (re)labeling, gap filling or smoothing, the file needs to be reprocessed to recalculate the skeleton data as follows:

1. Open the **Reprocessing dialog** by clicking the **Reprocessing** icon or **Reprocess** in the **Capture** menu.
2. Check the **Solve Skeleton** option. Make sure that other processing options that may alter the trajectories, e.g., **Track the measurement** and **Apply the current AIM models** are unchecked.
3. Press the **OK** button.

Export and streaming of skeleton data

You can export skeleton data for further processing in third-party software, such as MotionBuilder and Maya. The export formats are:

- FBX: a widely used format for exchange of 3D data for animation software, see chapter "Export to FBX file" on page 406
- TSV: a tab-separated text format, see chapter "Export to TSV format" on page 389
- MAT: Matlab data file, see chapter "Export to MAT format" on page 399

Real time streaming of skeleton data is supported by the QTM RT protocol. QTM connect applications are available on GitHub for MotionBuilder, Maya, Unity and Unreal. The streamed skeleton data can be scaled by using a scale factor, see chapter "Scale factor" on the previous page. For more information on how to stream skeleton data, see the documentation included in the respective QTM connect applications or the documentation on the RT Protocol included in the QTM installation.

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 187.

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 327. 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. Thenset the settings on the **2D tracking** page, see chapter "2D tracking" on page 187.
    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 68.

The following features 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.
- The keyboard shortcuts **J** and **Shift + J** can be used to jump to the next unidentified trajectory or part.

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 the 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.

4. Enable **Show labeled trajectory information** and **Show trajectory count** in the **3D view settings** for a quick overview of the quality of the labeling at the current frame.

Identification methods:

1. Use **Quick identification** to identify markers in the 3D view.
2. Drag and drop markers from the 3D view to the Labeled trajectories list or onto a trace in the 3D view.
3. 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

1. Use scrubbing (**Ctrl + drag**) and trace range zoom (**Shift + Mouse wheel**) features for browsing through the measurement.
2. 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.
3. Use the **Center on trajectory** tool (keyboard shortcut **C**) to center the view on the trajectory that you are interested in.
4. Use the **Jump to next trajectory/part** tool (keyboard shortcut **J** and **Shift + J**) to jump to and center on the next unidentified trajectory or part in time.
5. 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.
6. 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.
7. Use the **Cut trajectory trace** tool (keyboard shortcut **X**) 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 “How AIM identifies trajectories” on page 374.

Generating an AIM model

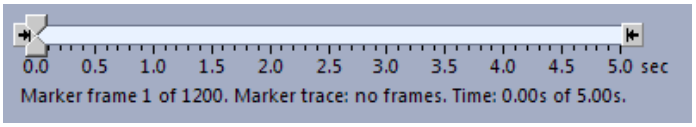
AIM needs a model file to work, which is generated from a measurement or multiple measurements with identified trajectories.

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)**

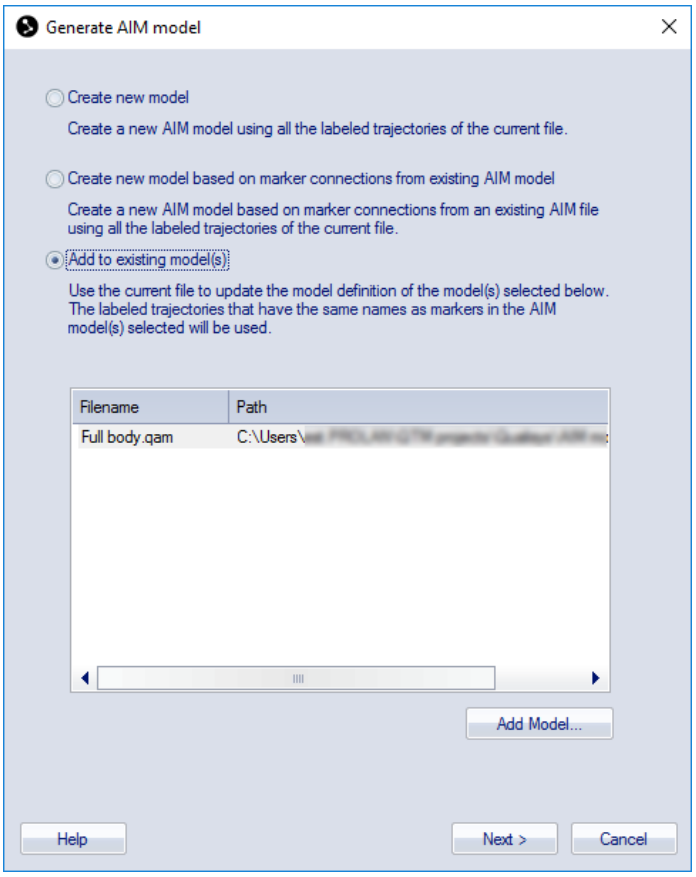
setting when you identified the file manually, because if you create a new model for each subject you will not use the full potential of the AIM functionality.

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 369: 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 to an AIM model, it becomes less important that the added files include a lot of movement.
     Note: Even if you want to make an AIM model for a static measurement it is important to have some movement in the first file. This is to make sure that the internal bone definitions are correct. However, the subsequent files can be static because then the definition is already correct.
   - 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.
   - In the first file used to generate an AIM model, it is recommended to not have any static markers together with the moving subject. Because it will make it very hard for AIM to find a solution. Only use an AIM model with static markers if the markers are always static.
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.

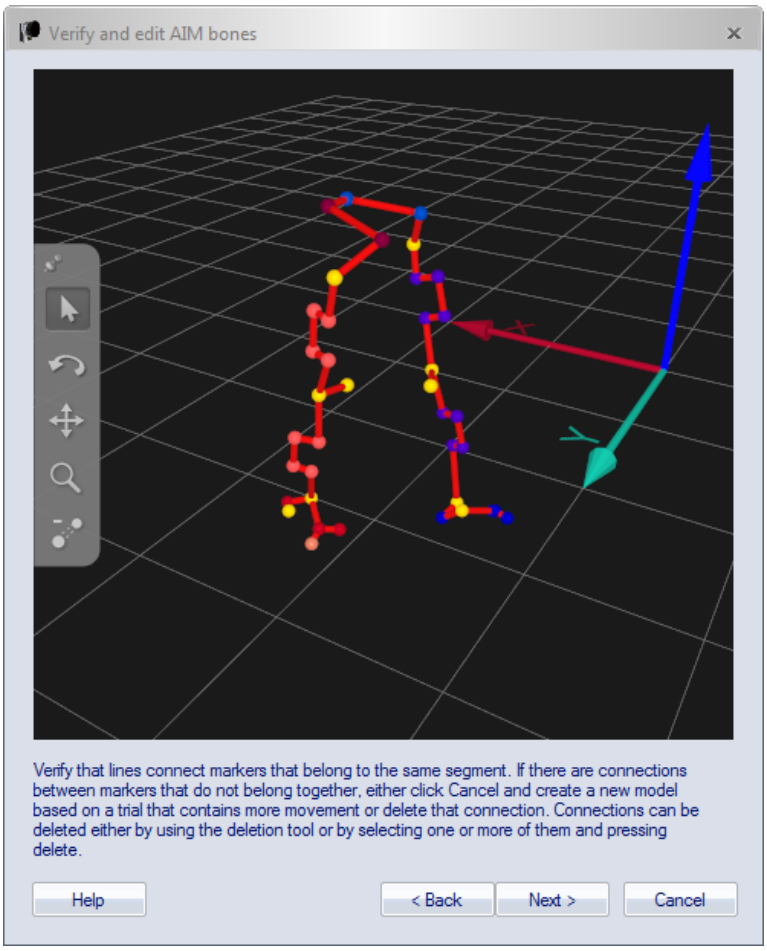


3. Click the **Generate model** icon on the AIM toolbar or click **Generate model** on the **AIM** menu.
4. Select whether you want to **Create new model** or **Add to existing model**. Use add to existing model if you already have an AIM model with the current marker setup.



Create new model

Click **Next** and then verify the AIM bones. It is very important that the bones are connected in the correct way so that not the wrong markers are connected. If the bones are wrong try to use the **Delete bones** tool in the toolbar to remove the bones that are wrong. For more information about the AIM bones see section “How to verify and edit AIM bones” on page 370.



Then select the directory where you want to save the model. The default location is to save it in the AIM folder of the project. Enter a **File name** for the model and click OK. The model is generated automatically.

There can be two warnings when generating AIM models related to static markers.

a. First if all labeled markers are too static there is a warning because a static measurement can lead to erroneous internal bones in the AIM model, for example two knees can be joined. This will then be used even if the added files are with a movement. You should only use an AIM model with static markers if the markers are always static.

b. The other warning is if some of the labeled markers in the file are static while others are moving. The problem is the same as above that you will get internal bones in AIM that makes it harder to apply the model. If you need static markers you can either create a separate AIM model for those markers or have empty labels in the first AIM model and identify them manually.

Create new model based on marker connections from existing AIM model
Use this option to create a new AIM model based on the structure of an existing model, preserving its AIM bones and its visual properties of the trajectories (colors and bones). The data of the existing AIM model is ignored, so the new AIM model is based on the movements of the current measurement only. The labels of the measurement should fully correspond to the labels of the existing AIM model. This option consists of the following steps.

a. Select an existing AIM model from the file dialog. If the labels of the selected model do not correspond to those of the measurement, AIM generation will fail.

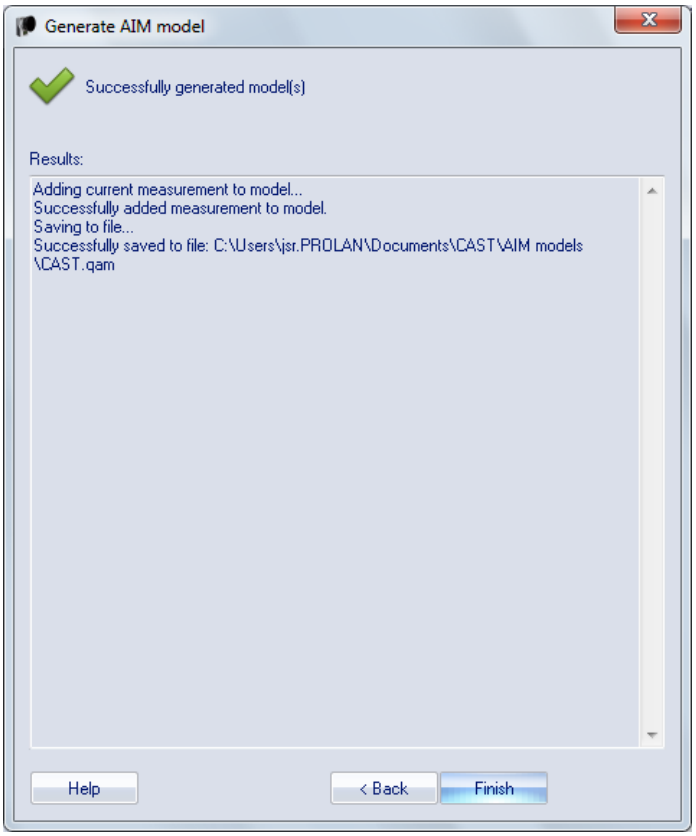
b. Review the AIM model in the AIM model visualization window. Note that the AIM bones cannot be edited.

c. Specify the file name of the new AIM model.

Add to existing model
Select a model or models from the list to which you want to add the movement in the current file. This option is 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. The previous AIM model is automatically saved in the AIM models/Backup folder.

- 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.
- If you only add to one AIM model you will get a dialog that displays the AIM bones of the AIM model that is added to. If the AIM bones are placed incorrectly in the model it is best to make a new AIM model and start training that one instead.

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. For example you will get a warning if the first file is static or if the added file is too different and in those cases you can choose to abort the process.



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 to an AIM model, it becomes less important that the added files include a lot of movement. You can verify that the movement is large enough by looking at the AIM bones, see section “How to verify and edit AIM bones” on the next page.

  Note: Even if you want to make an AIM model for a static measurement it is important to have some movement in the first file. This is to make sure that the internal bone definitions are correct. However, the subsequent files can be static because then the definition is already correct.

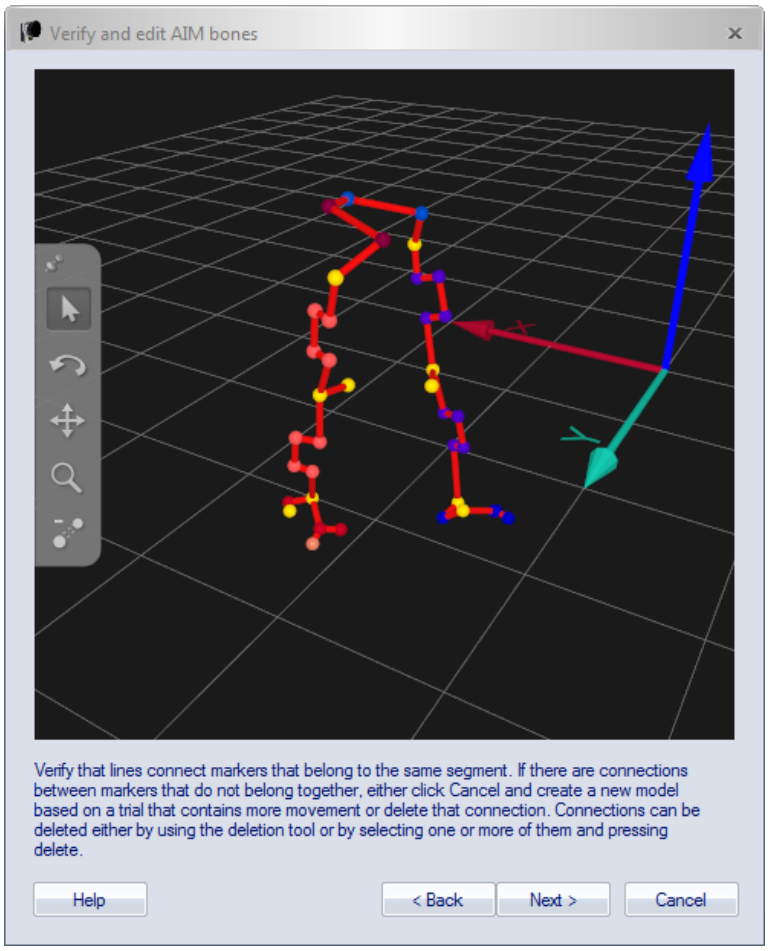
- Make sure that the trajectories have the correct identity throughout the file. You can select a smaller measurement range to delete the incorrect parts if you do not want to identify the whole file.
- The colors of the trajectories and any bones between them are also saved in the model. For example 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 in the same measurement it is recommended to make an AIM model for each subject, see “AIM models for multiple subjects in the same measurement” on page 375.

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 72.
    - 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

How to verify and edit AIM bones

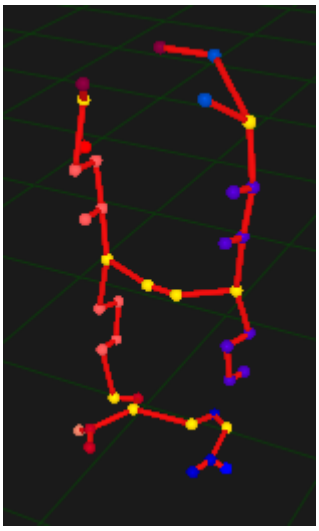
When an AIM model is created there are AIM bones created automatically between the markers that have least movement between them throughout the file. Because these AIM bones are then kept even if you add more measurements it is important that they are correct. Therefore there is a step when creating a new AIM model where you can verify and edit the AIM bones.



- The AIM bones are displayed as red bones between the markers.
- There is no relationship between the AIM bones and bones created by the user.
- The number of AIM bones are as few as possible.
- You can rotate, translate and zoom the 3D image of the AIM bones.
- You can delete AIM bones with the **Delete bone** tool, see below.
- The frame used for displaying the AIM model is the first frame that include all or as many as possible of the markers.

To verify that the AIM bones are correct follow these instructions:

1. Check that all of the markers are displayed. If there are missing markers it means that there is no frame with all of the markers. It is recommended to use another file where all of the markers are available in at least one frame.
2. Check that the AIM bones are connected between the correct segments. For example if you are creating a model of a person that has been standing still then the knees and feet may be connected with AIM bones, see image below. Then use the **Delete bone** tool to remove the bones or select the bone and press Delete. Every time a bone is deleted AIM will look for a new solution, so you may have to delete several bones before you have the correct solution. This is especially true if the measurement is static, then it is often better to create a new file with more movement.



3. The AIM bones may not look as good as your user created bones, but that does not necessarily mean that it is wrong. For example the AIM bone between a head and the body can look strange if the marker nearest to the head is on the shoulder.

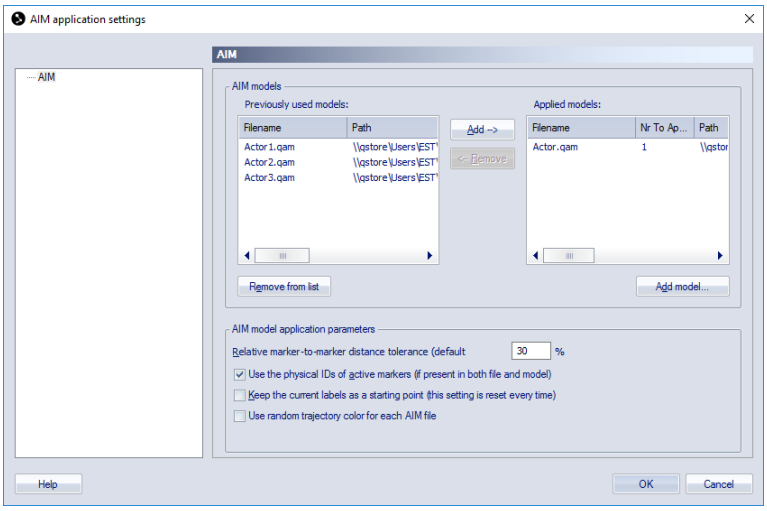
Applying an AIM model

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.

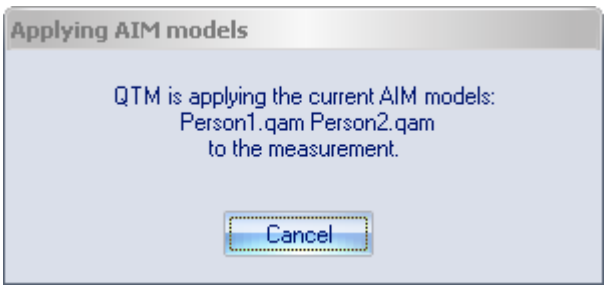
   Note: The AIM settings are copied from Project options, if you want to use the AIM settings and models saved in the file you need to reprocess the file and select from **measurement** as source, see chapter “Reprocessing a file” on page 327.



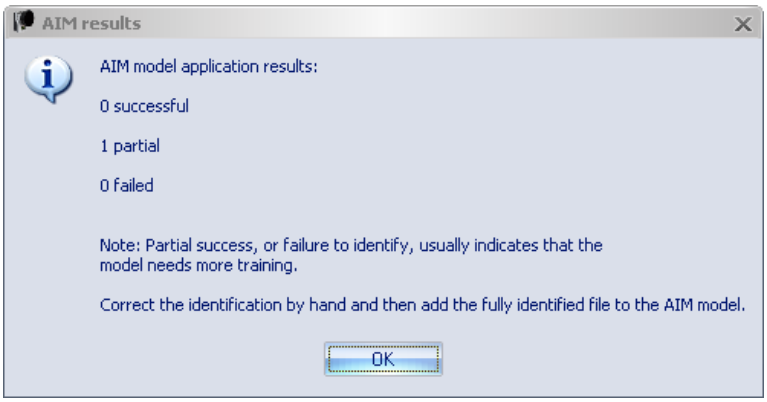
4. Check that the **Applied models** are correct. It is possible to apply several AIM models to the same file, or to apply the same AIM model to multiple actors. In the latter case, set **Nr To Apply** to the number of actors. For more information see "AIM models for multiple subjects in the same measurement" on page 375 .

   You can also change the **AIM model application parameters**, see "AIM model application parameters" on page 194. However if you add measurements to the AIM model you usually do 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 doesn't completely match the captured motion. Then it is recommended to manually identify the file and then add it to the existing AIM model, see “Generating an AIM model” on page 364.

Note: When you get a partial solution it is recommended to retrack the file before trying to apply a modified AIM model, since the cuttings made by AIM can make it harder to apply AIM the next time.

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, for more information about AIM see chapter “How AIM identifies trajectories” below.
   - AIM will join parts of trajectories that are within the AIM model definition and also split the trajectory if it is too much outside the AIM model definition. Therefore if there are some trajectories that are not identified in a part of the measurement it probably did not match the model in that part. The solution is to identify the markers and then add the file to the AIM model.
   - AIM sometimes cannot identify a trajectory if a nearby trajectory is missing during that part of the measurement.
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.

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 needs to have movement in the first file used to create the model. Otherwise AIM will not identify the correct bones in the AIM model, since the bones are based on the shortest possible distances between markers.
3. AIM looks for the best solution in each frame and then joins the solutions. This means that AIM will join and split trajectories depending on the AIM

model so that you get the best match. The measurement subject can leave and enter the volume and AIM will continue to join the trajectories.

4. 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 in the same measurement

You can apply AIM for automatic labeling of multiple subjects at the same time. The following scenarios can be distinguished:

- Tracking of subjects with different marker configurations and labels. This requires a unique AIM model for each subject, which should be present in the **Applied models** list in the AIM dialog or settings page.
- Tracking of subjects with similar marker configurations (e.g. multiple actors with same marker set). In this situation the same AIM model can be applied multiple times by specifying **Nr to Apply** in the AIM dialog or settings page.
- A combination of the above.

For more information about how to create and apply AIM models when capturing multiple subjects, see the chapters below.

Unique subjects

For a unique identification of 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 364 and “Applying an AIM model ” on page 372. If you already have a generic AIM model, which you want to use to create new AIM models for unique subjects, see chapter “Use generic AIM model to create unique models” on the next page.

1. Make sure that each subject has a unique marker configuration. If you have similar types of subjects (e.g. multiple actors), you can use a different marker pattern to distinguish the subjects, for example by placing four markers on the chest in different patterns.
2. Make sure that the label names of the subjects are different so that QTM can identify the labels when you add measurements to the AIM models.
3. Create an AIM model for each subject. This can be done per subject as described in “Generating an AIM model” on page 364. Alternatively, you can generate the AIM models using a single measurement with multiple subjects as follows.
    a. Make a Range Of Motion (ROM) measurement with all subjects simultaneously.
    b. Label the markers per subject as described in “Manual identification of trajectories” on page 362.

c. Select the trajectories of one subject in the 3D view or the trajectory list, right-click on the selection and click **Generate AIM model from selected trajectories...**.

d. Generate the AIM model as described in chapter "Generating an AIM model" on page 364

e. Repeat step b. to d. for every subject.

4. To apply multiple AIM models to a measurement you must 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.

5. You can add data to multiple AIM models from a single measurement by selecting the applicable AIM models under the **Add to existing model(s)** option. The data will then be added to the AIM model(s) with matching label names.

Similar subjects

When tracking similar subjects with the same marker configuration, you can use one generic AIM model and apply it multiple times. This is done as follows:

1. Add the AIM model to the **Applied models** list on the **AIM** page in the **Project options** dialog.
2. Set **Nr To Apply** to the number of subjects to be tracked simultaneously.
3. Optionally, enable **Use random trajectory color for each AIM file**. This will help to distinguish the subjects from one another.
4. Start tracking.

Note: It is not possible to uniquely identify subjects when applying a generic AIM model multiple times. The order of the subjects may be different for every measurement.

Use generic AIM model to create unique models

If you are measuring multiple subjects with the same marker configuration you can create specific AIM models for each subject based on a generic AIM model. The advantage is that this allows for identification of individual subjects.

1. First, follow the steps in "Similar subjects" above

   a. Add the AIM model to the **Applied models** list on the **AIM** page in the **Project options** dialog.

   b. Set **Nr To Apply** to the number of subjects to be tracked simultaneously.

   c. Optionally, enable **Use random trajectory color for each AIM file**. This will help to distinguish the subjects from one another.

   d. Make a measurement with multiple subjects.

2. Select the trajectories of one subject in the 3D view or the trajectory list.
3. Optionally, a prefix can be added to the selected labels by right-clicking on the selection and clicking **Add prefix to selection...**. This can be helpful to distinguish subjects from each other, especially when using the same colors scheme.

4. Right-click on the selection and click **Generate AIM model from selected trajectories...**.
5. Generate the AIM model as described in chapter “Generating an AIM model” on page 364.
6. Repeat steps 2. to 4. for each subject.
7. Start tracking.

Editing of trajectories

The **Trajectory Editor** can be used to view and edit trajectory data. Manual editing trajectory data should be the last step in the processing of trajectories. Before editing trajectory data, it is important to make sure that the quality of the tracking is optimal, and that the identification of trajectories is correct and as complete as possible. Remaining irregularities of trajectories can be edited with the **Trajectory Editor**.

The main functions of the **Trajectory Editor** are:

- locating and filling gaps, and
- detecting and smoothing artifacts.

You can open the **Trajectory Editor** window in the following ways:

- by clicking **Trajectory Editor** in the **View** menu,
- by clicking the **Trajectory Editor** button on the **Trajectory toolbar**,
- via the keyboard shortcut **Ctrl + T**.

The functions and lay-out of the **Trajectory Editor** is described in chapter “Trajectory Editor window” on page 79.

The following chapters describe in more detail how to use the **Trajectory Editor**.

Gaps

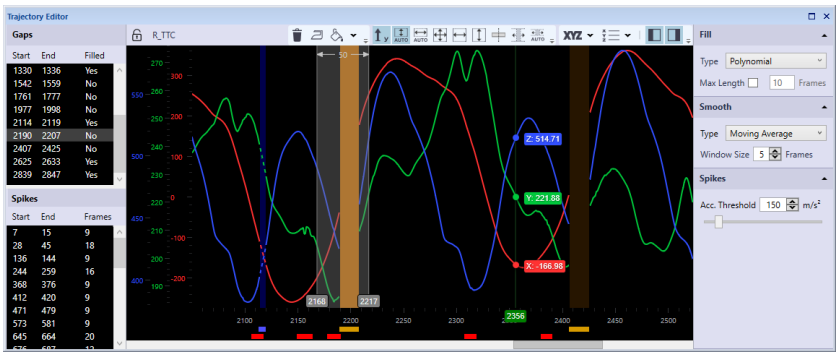
Gaps are to be understood as missing parts within a trajectory. Two common types of gaps are:

1. Gaps due to missing 3D data, for example in case of occlusions.
2. Gaps due to incomplete identification.

The second type of gaps can generally be kept to a minimum by improving the AIM model. In some cases, manual identification may be needed to add unidentified parts to the trajectory. Remaining gaps can be detected and filled in the **Trajectory Editor**.

The following chapters describe the gap filling facilities in the **Trajectory Editor**.

Identification and selection of gaps

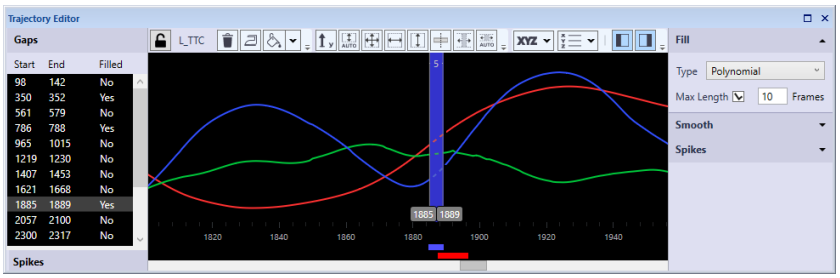


Gaps are indicated in the plot area by a brown area and an amber indicator below the time axis at the frames where data is missing. The gaps are also listed in the **Gaps panel** in the **Points of Interest sidebar**.

You can select a gap in the graph by clicking on the gap area. The selection range is then set from the start to the end of the gap, and the gap area will be highlighted.

Alternatively, you can select gaps in the **Gaps panel** in the **Points of Interest sidebar**. By holding the **Shift** key you can select multiple gaps. The selection range is then set from the start of the first gap to the end of the last gap. When double clicking on a gap in the **Gaps panel**, the current frame will be moved to the start of the selection.

Filling of gaps



To fill gaps follow these steps:

1. Select the trajectory you want to edit. You can select the trajectory in one of the **Trajectory info** windows, or by clicking on the marker in the **3D view** window.
2. Select a gap or a frame range that includes the gaps you want to fill.
3. Press the **Fill** button.

The gaps included in the selected range will then be filled using the **Type** specified in the **Fill settings**. The filled parts are indicated in the plot area by a dark blue area and a blue line below the time axis at the gap-filled frames, and the data series is shown as a dashed line.

The available fill types can be divided into two categories. **Linear**, **Polynomial**, **Relational** and **Kinematic** are gap fill types that *interpolate* the data between the respective edges of the trajectory. Polynomial gap fill can only be applied when there is trajectory data on both sides of the gap. Linear and relational gap fill can also be used for extrapolation for gaps at the start or end of the capture. The filled parts of the trajectory are indicated as type *Gap-filled* in the **Trajectory info** window. **Static** and **Virtual** are *virtual fill types* that are independent of the data at the edges of the gap. These types can also be applied when there is no surrounding data, for example to

empty trajectories or to gaps at the start or the end of the capture. The filled parts of the trajectory are indicated as type *Virtual* in the **Trajectory info** window.

Static
Gaps are filled with fixed values for X, Y and Z as specified.

Linear
Gaps are filled by a linear interpolation between the respective edges of the trajectory. If the gap is at the beginning or the end of a capture, the gap will be filled with a constant value (first or last data value after or before the gap, respectively). Options:

Max Length: Check to apply a maximum length for linear gap filling.

Frames: Specify maximum number of frames for linear gap filling.

Polynomial
Gaps are filled by a cubic polynomial interpolation between the respective edges of the trajectory. Polynomial gap fill requires that the parts before and after the gap have a length of at least two frames. Options:

Max Length: Check to apply a maximum length for polynomial gap filling.

Frames: Specify maximum number of frames for polynomial gap filling.

Relational
Gaps are filled based on the movement of surrounding markers. The user specifies one, two or three context markers, which are used to define a local coordinate system (LCS). The filled trajectory consists of a linear interpolation in the LCS, which is then transformed to the global coordinate system. If the gap is at the beginning or end of a capture, the filled part will be extrapolated. The options can be selected from a drop down list. Alternatively, you can drag and drop trajectories on the field if the trajectory is locked. The following options are available:

Origin (required): Marker defining the origin of the LCS, preferably a marker close to the target marker. If only **Origin** is defined, the filled trajectory will be entirely translational.

X Axis: Marker defining the primary axis of the LCS. Preferably, the movement of the primary axis should be strongly correlated with that of the target marker. If the target marker is on the same line as the **Origin** and the **X Axis** markers, it should be sufficient to specify only these two context markers.

XY Plane: Marker defining the secondary axis of the LCS, fixating its full orientation.

Rigid body: Check this option in case the three context markers are part of a rigid structure. If checked, the pose of the LCS will be calculated by means of a rigid body fit. The definition of the rigid body will be based on the average of the relative configuration of the context markers across the gap. This option can only be checked if all three context markers are specified.

Virtual
Gaps are filled based on the movement of surrounding markers. Similar to **Relational** gap fill, except that the filled part is independent of surrounding trajectory data. The user specifies one, two or three context markers, which are used to define a local coordinate system (LCS). The filled trajectory represents the movement of the origin of the LCS, with an optional offset specified by the user. The options can be selected from a drop down list. Alternatively, you can drag

and drop trajectories on the field if the trajectory is locked. The following options are available:

Origin (required): Marker defining the origin of the LCS, preferably a marker close to the target marker. If only **Origin** is defined, the filled trajectory will be entirely translational.

X Axis: Marker defining the primary axis of the LCS. Preferably, the movement of the primary axis should be strongly correlated with that of the target marker. If the target marker is on the same line as the **Origin** and the **X Axis** markers, it should be sufficient to specify only these two context markers.

XY Plane: Marker defining the secondary axis of the LCS, fixating its full orientation.

Rigid Body: Check this option in case the three context markers are part of a rigid structure. If checked, the pose of the LCS will be calculated by means of a rigid body fit. The definition of the rigid body will be based on the average of the relative configuration of the context markers across the gap. This option can only be checked if all three context markers are specified.

Offset: Apply an offset to the virtual point, relative to the origin of the LCS.

X, Y, Z: Offset values of X, Y and Z in mm or percent.

Relative Offset (%): check to specify offset in percent relative to the distance between the **Origin** and **X Axis** markers.

Kinematic
Kinematic gap fill of markers associated with skeleton segments or rigid bodies based on current skeleton or 6DOF data.

Note: In some cases there may be a spike in the trajectory just before or after a gap. Deleting such artifacts before gap filling can improve the quality of the filled part, in particular for the polynomial method.

Spikes

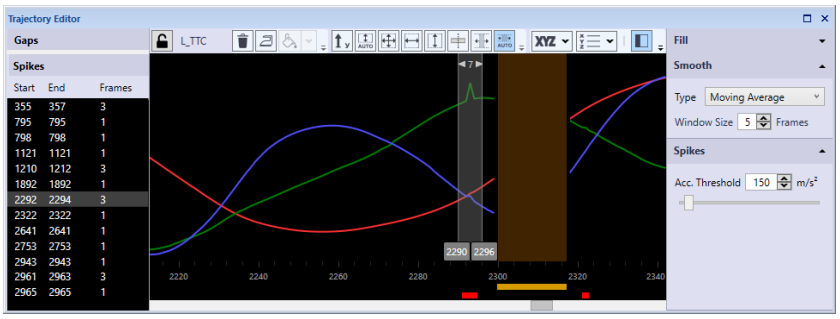
Spikes are to be understood as discontinuities between consecutive frames within a trajectory. The **Trajectory Editor** can be used as a tool to detect spikes. Two common types of spikes are due to:

1. Tracking artifacts, for example due to occlusions or partial occlusions of markers in one or more cameras.
2. Labeling artifacts, for example due to swapping of adjacent markers.

Tracking artifacts can generally be kept to a minimum by optimizing camera setup, camera settings for the best possible visibility and distinction of markers, as well as by optimizing the tracking parameters. Labeling artifacts can be avoided by using an appropriately trained AIM model (see chapter “Automatic Identification of Markers (AIM)” on page 364), or repaired by swapping parts (see chapter “Swap parts” on page 73). As a final step in the processing of data, the resulting spikes can be edited in the **Trajectory Editor**.

The following chapters describe how to detect and smooth spikes in the **Trajectory Editor**.

Detection and selection of spikes



The detection of spikes is based on an acceleration threshold, which can be adjusted in the **Settings Sidebar** of the **Trajectory Editor** window.

Spikes are indicated in the plot area by a red indicator below the time axis. The spikes are also listed in the **Spikes panel** in the **Points of Interest sidebar**.

You can select a spike by clicking on the corresponding indicator below the time axis. The selection range is then set from four frames before the start to four frames after the end of the spike.

Alternatively, you can select spikes in the **Spikes panel** in the **Points of Interest sidebar**. By holding the **Shift** key you can select multiple spikes. The selection range is then set from four frames before the start of the first spike to four frames after the end of the last spike. When double clicking on a spike in the **Spikes panel**, the current frame will be moved to the start of the selection.

Smoothing

To smooth spikes follow these steps:

1. Select the trajectory you want to edit. You can select the trajectory in one of the **Trajectory info** windows, or by clicking on the marker in the **3D view** window.
2. Select a spike or a frame range that includes the spikes you want to smooth.
3. Press the **Smooth** button.

The spikes included in the selected range will then be smoothed using the **Type** specified in the **Smooth settings**. The edited parts of the trajectory are indicated as type *Edited* in the **Trajectory info** window.

Note: When applying smoothing to a selection, the whole selected range will be smoothed, not only the detected spikes.

The following smoothing types are available:

Moving Average
Smoothing by means of a unweighted moving average across a centered window. The **Window Size** can be set by the user. The **Moving Average** type is most suitable for smoothing of local spikes.

Butterworth
Smoothing by means of a fourth order Butterworth low-pass filter. The **Butterworth** type is most suitable for reduction of high-frequency noise across large frame ranges. The available options are:

Cutoff: Cutoff frequency specifying the pass band and the stop band of the filter. As a rule of thumb, the cutoff frequency should be a factor 2-3 higher than the highest frequency of interest.

Alternatively, spikes can be removed by means of deleting and gap filling. To delete a spike, you can select a frame range and delete it by pressing the **Delete** button, or by pressing the **Delete** key on the keyboard.

Note: The minimum frame range for the Butterworth filter is 7 frames. This minimum also applies to the size of parts in case the selected frame range includes gaps. When the selected frame range includes parts that are smaller than 7 frames, the Butterworth filter cannot be applied.

Note: The Butterworth filter is applied in forward and reverse direction to compensate for the latency of the filter. The order and cut-off frequency are adjusted so that the effect of the filter is in accordance with the specified order and cutoff frequency when applying the filter in one direction.

Adding virtual trajectories

You can add a virtual trajectory in the following ways.

1. Select multiple trajectories and create a virtual trajectory at the geometric average of the selected trajectories via the **Trajectory info window menu**.
   a. Select one or more trajectories and right click on the selection in the **Trajectory info** window or in the **3D View** window.
   b. In the context menu, click **Add new trajectory** > **Virtual (Average of selected trajectories)**. This will create a new trajectory at the geometric average of the selected trajectories.
   c. Name the new trajectory.
2. Create a virtual trajectory via the **Trajectory Editor**.
   a. Create a new empty trajectory via **Trajectory info window menu** or by pressing the Insert (**Ins**) key and name it.
   b. Select the new trajectory.
   c. Open the **Trajectory Editor** window, if it is not already visible.
   d. In the **Settings sidebar** of the **Trajectory Editor** window, choose fill type **Constant** or **Virtual**.
   e. Specify the fill options:

      For type **Constant**, specify X, Y and Z coordinates. This will result in a static virtual trajectory at position X, Y and Z in the global coordinate system.

      For type **Virtual** select the trajectories on which the virtual trajectory should be based. The options can be selected from a drop down list. Alternatively, you can drag and drop trajectories on the field if the trajectory is locked. By default, the virtual trajectory will correspond to the trajectory selected as **Origin**. By specifying the X, Y, and Z values under Offset, you can move the virtual trajectory to a different position, relative to the local coordinate system as specified by the fill options. See chapter “Filling of gaps” on page 378 for more information about the options.

How to use 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
You can use an external trigger to generate an events during a measurement. When using the Qualisys trigger button, 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 **Synchronization** page.

The event functionality is activated by default but can be changed with the **Generate event** setting for the external trigger on the **Synchronization** page, see chapter "Trigger ports" on page 156.

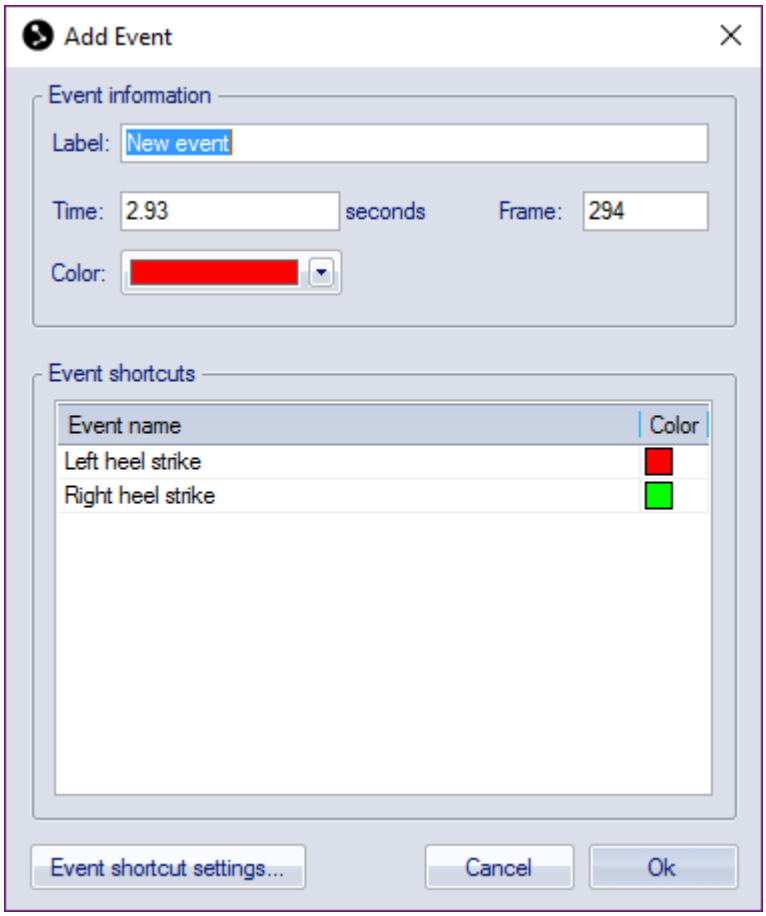
Note: Events cannot be created if you have selected to stop on the external trigger.

When the events are created during a capture it will be stored in the QTM file as **Trigger** or **Trigger start** event. The **Trigger start** event is only used for start event of a pretrigger measurement. You can change the default color of these events on the **Synchronization** page in **Project options**, see chapter "Synchronization" on page 154.

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
A manual event can be created with the **Add event** button or by using the keyboard shortcut **Ctrl+E**. The events can be created with the button both during RT/capture and after a measurement. In file mode, events can even be added via the **Events** submenu by right-clicking in the **Timeline control** bar.

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**, **Frame** and **Color**.

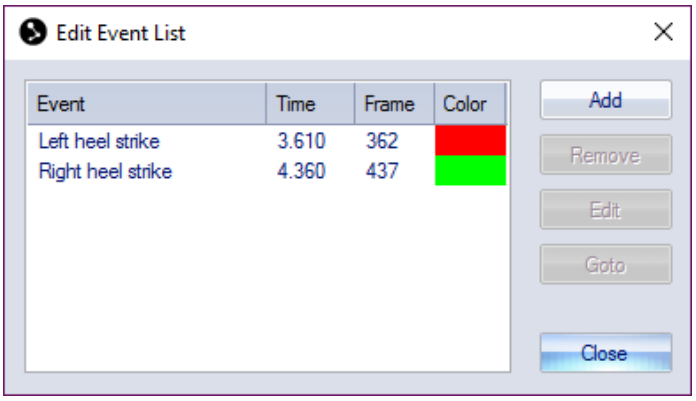
You can also use event shortcuts in the **Add event** dialog to create the events. Double-click on an event shortcut in the list to load the label name and color. To edit the shortcuts click on **Edit event shortcuts**, that opens the **Events** page in the **Project options** dialog, see chapter “Events” on page 252.

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**.
- You can go to next and previous event in the file with **Page Down** respectively **Page Up**.
- 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
Go to the frame of the event in the file.

Exporting events

The events can be exported to other programs with the C3D, TSV and MAT 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 need to activate the **Include events** option to export the events. For more information about the TSV format, see chapter “Motion data (.tsv)” on page 390.

For the MAT export you need to activate the **Events** option to export the events. For more information about the MAT format, see chapter “MAT file format” on page 399.

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 211. 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

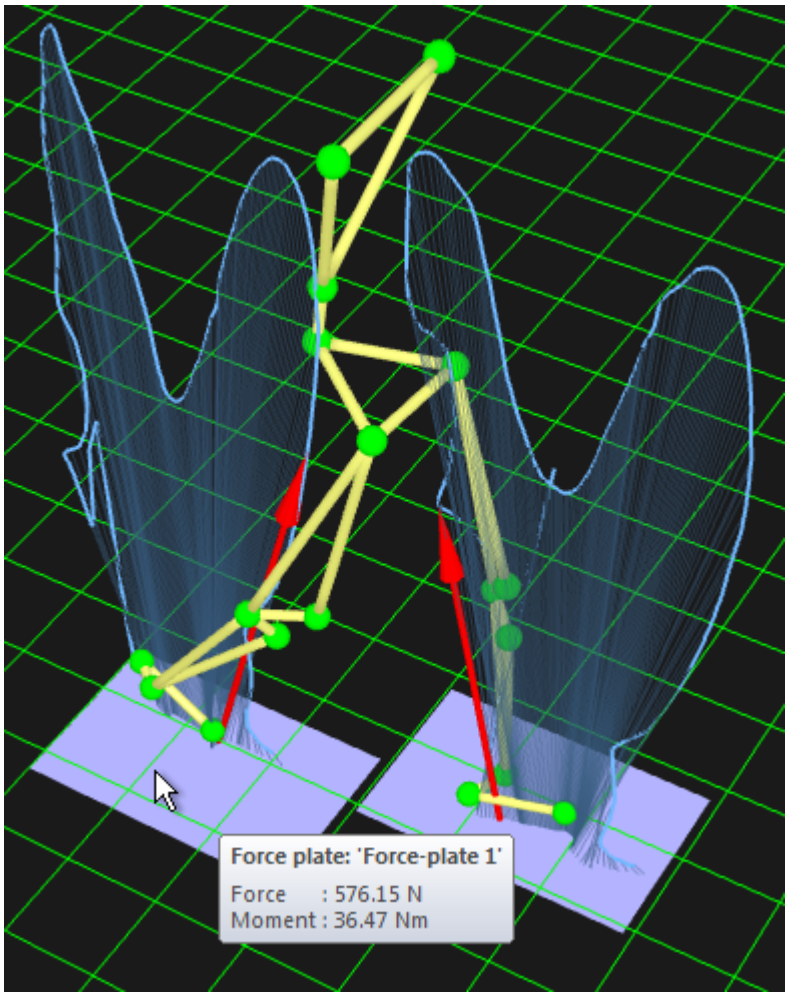
Just before the Start capture dialog is opened before a capture

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 92.



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 global 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 224.

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 to other

applications” on page 389. 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

Exported measurement data can be analyzed in other applications. The data can be exported to the following different formats:

TSV format, see chapter “Export to TSV format” below.

C3D format, see chapter “Export to C3D format” on page 397.

Mat format, see chapter “Export to MAT format” on page 399.

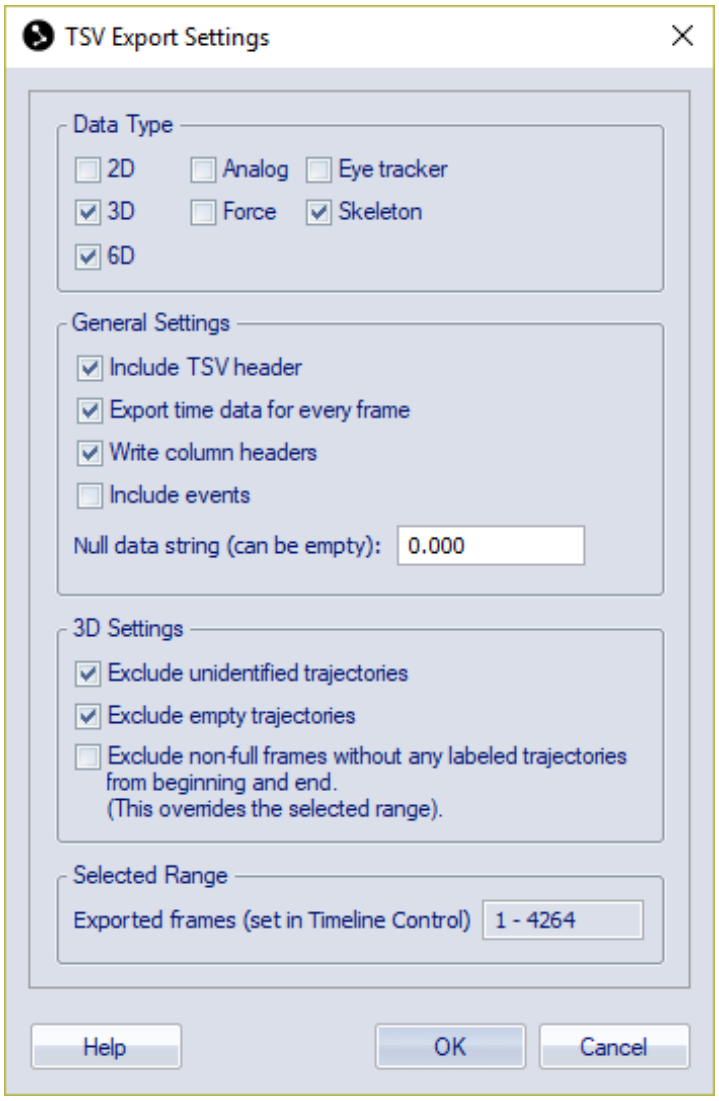
AVI file, see chapter “Export to AVI file” on page 404.

FBX format, see chapter “Export to FBX file” on page 406.

The data export can be done either as a processing step or manually on the **File/Export** menu. 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 181.

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 231. 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

The following data types can be exported as TSV (Tab Separated Values) from QTM:

- 3D and 2D motion data
- 6DOF data
- Skeleton data
- Analog data
- Force data
- Eye tracker data
- Skeleton data

For a detailed description of the respective file formats, see the chapters below.

Each data type is exported as a separate file. By default, TSV files have the same name as the QTM file with an additional suffix, depending on the data type. In case there are multiple devices (e.g. analog devices, force plates), one file per device is exported with an index added to the suffix.

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 and ticks from when the computer was started.

Note: The time stamp is recalculated when trimming the file.

WARNING: The time stamps indicate the start of the capture according to the QTM software. The time stamps may not exactly correspond to the first frame of the capture and it is discouraged to use them for synchronization with data from other devices connected to the same or another synchronized computer.

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 which you can use to add more information to the file.

When the **Export time data for every frame** option is checked, two columns will be added containing time data. The first column contains the frame number and the second contains the time in seconds relative to the start of the capture. If the camera frames contain timestamps (SMPTE or IRIG), a third column with timestamp data is added. The format of the timestamp string depends on the type of timestamp used.

When the **Write column header** option is checked, a header is added above each column describing the contents of that column.

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.

The header part is only written when the option **Include TSV header** is checked. The header part contains the following elements.

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.

BODY_FILTERS
Tab-separated list with the names of the used filter presets for the respective rigid bodies.

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 202.

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 “Appendix F: Rotation angle calculations in QTM”.

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 which you can use to add more information to the file.

When the **Export time data for every frame** option is checked, two columns will be added containing time data. The first column contains the frame number and the second contains the time in seconds relative to the start of the capture. If the camera frames contain timestamps (SMPTE, IRIG or Camera time), a third column with timestamp data is added. The format of the timestamp string depends on the type of timestamp used.

When the **Write column header** option is checked, a header is added 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

Skeleton data (_s_name.tsv)

The skeleton data files contain the skeleton data included in the capture. There will be one file for each skeleton in the capture. The file name has suffix _s_<skeleton name>, where the last part corresponds to the name of the respective skeletons in the **Skeleton Solver** page in **Project Options**. The file contain 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.

The header part is only written when the option **Include TSV header** is checked. The header part contains the following elements.

NO_OF_FRAMES
Total number of frames in the exported file.

NO_OF_CAMERAS
Number of cameras used in the motion capture.

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.

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.

REFERENCE
Reference for used for the skeleton data. Global: all segment positions and rotations are expressed relative to the global coordinate system. Local: all segment positions and rotations except the Hips segment are relative to their respective parent segment.

When the **Write column header** option is checked, a header line is added above each column describing the contents of that column. The column headings for each segment are:

Segment name
Name of the respective skeleton segments. There will be no data in the column below.

X, Y and Z
Segment position data in mm.

QX, QY, QZ, QW
Segment rotation data in global coordinate system expressed as quaternions.

The data is stored in tab-separated columns, where each row represents a frame. The format of the data part depends on the chosen options.

When **Write column header** is **disabled** there are 7 columns per segment containing the position and orientation data. Columns for the respective segments are appended without any extra space, making up a total of 154 columns for all 22 segments.

When **Write column header** is **enabled** there are 8 columns per segment. The first column below the segment name is empty, the remaining 7 columns contain the position and orientation data for the segment. The total number of columns is 176 for all 22 segments.

When the **Export time data for every frame** option is checked, two columns will be added containing time data. The first column contains the frame number and the second contains the time in seconds relative to the start of the capture. If the camera frames contain timestamps (SMPTE, IRIG or Camera time), a third column with timestamp data is added. The format of the timestamp string depends on the type of timestamp used.

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. If you have a sensor on the EMG that gives you other data than V then it is in the SI unit of that type of data, except for accelerometers that are in g.

There are two options in the TSV export which you can use to add more information to the file.

When the **Export time data for every frame** option is checked, two columns will be added containing time data. The first column contains the frame number and the second contains the time in seconds relative to the start of the capture.

When the **Write column header** option is checked, a header is added 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. The file names end with _f and are indexed with the number of the force plate in the same order they appear on the **Force data** page (_f_1, _f_2 and so on).

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
Information about the coordinate system of the force plate in the file. It can be either **Force data in local (force plate) coordinates** or **Force data in world (lab) coordinates**.

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 (in mm) 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 (in mm) 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 (in mm) 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 (in mm) of the bottom right corner when looking at the internal force plate coordinate system. The position is in the measurement coordinate system.

FORCE_PLATE_OFFSET_X, **FORCE_PLATE_OFFSET_Y** and **FORCE_PLATE_OFFSET_Z**
Offset (in mm) applied to the force plate position. X, Y and Z are defined in the internal coordinate system of the force plate.

FORCE_PLATER_LENGTH
The length of the force plate (in mm).

FORCE_PLATE_WIDTH
The width of the force plate (in mm).

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. The data is in N (forces), Nm (moments) and mm (COP).

There are two options in the TSV export which you can use to add more information to the file.

When the **Export time data for every frame** option is checked, two columns will be added containing time data. The first column contains the frame number and the second contains the time in seconds relative to the start of the capture.

When the **Write column header** option is checked, a header is added above each column describing the contents of that column.

Eye tracker data (_g.tsv, _e.tsv)

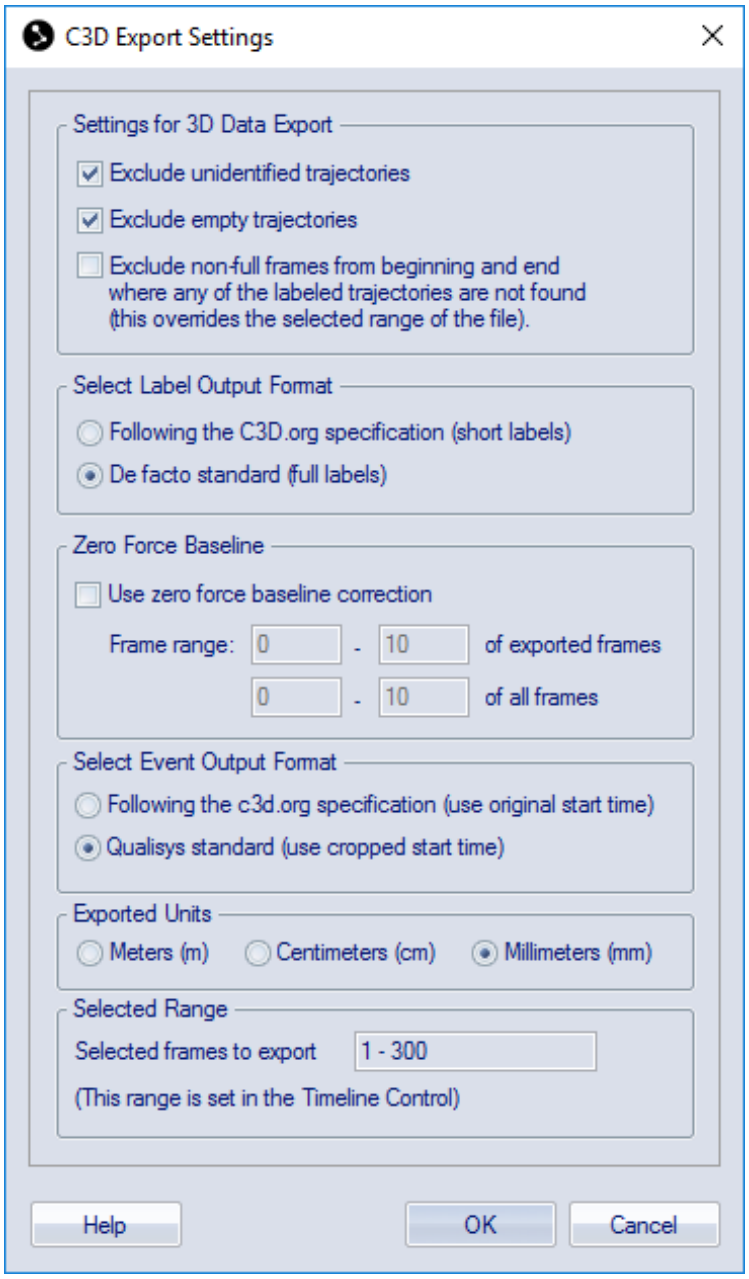
The eye tracker data files contain the data of eye tracking devices included in the capture. The data exported depends on the eye tracking device used. For more information about the export of specific eye tracker devices, refer to the detailed information of the device.

- For Tobii, see “ Process and export Tobii gaze vector data” on page E - 28.
- For SMI, see “Process and export SMI gaze vector data” on page E - 18.
- For Dikablis, see “Process and export Dikablis gaze vector data” on page E - 9

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 next 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 233. 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 **Zero baseline** option is only used on force data and its frames are 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.

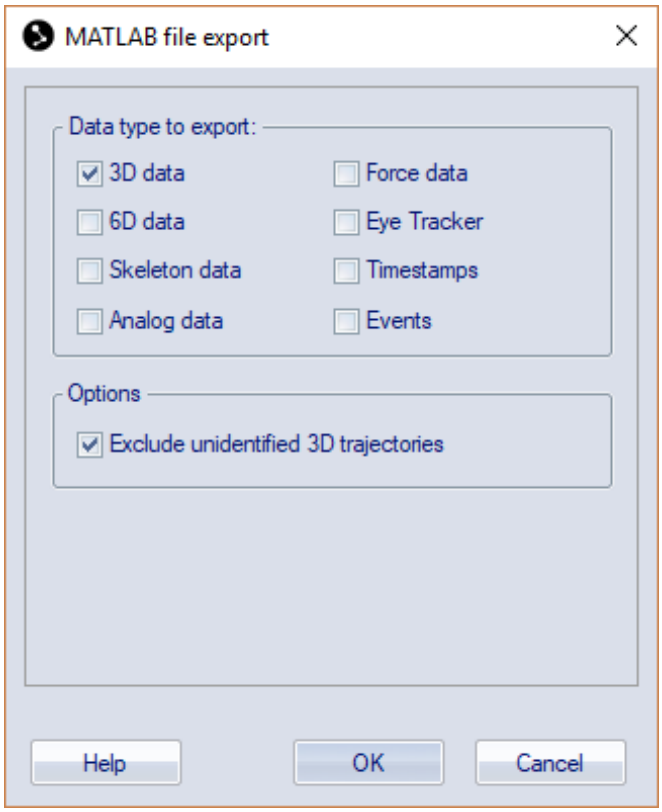
One limit of the C3D format is that the analog frequency must be the same for all analog data and a multiple of the marker frequency. If this is not the case then the analog data will be resampled at the multiple equal to or higher than the highest analog frequency in the measurement. For example if 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 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 234. For further information about the file format see chapter "MAT file format" below.



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. If the file name does not start with an English letter, a prefix qtm_ will be added to the name of the structure array.

The struct array contains different fields depending on if the file includes 3D, 6DOF, analog (including EMG), force, Eye tracker, SMPTE timecode and events data. The fields and their contents are described below:

File
File name and directory path.

Timestamp
Time when measurement was started. In date format YYYY-MM-DD, HH:MM:SS.SSS, followed by a tab character and the timestamp in seconds and ticks from when the computer was started.

Note: The time stamp is recalculated when trimming the file.

WARNING: The time stamps indicate the start of the capture according to the QTM software. The time stamps may not exactly correspond to the first frame of the capture and it is discouraged to use them for synchronization with data from other devices connected to the same or another synchronized computer.

StartFrame
The measurement start frame number.

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 with fields **Labeled** for labeled markers and, optionally **Unidentified** for unidentified markers. These fields are structs with the following 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.

Type
Type specification of the markers per frame. The data is given in a matrix with the dimensions: Trajectories*Frames. The types have values 0-4, which indicate: Missing=0, Measured=1, Gap-filled=2, Virtual=3, Edited=4.

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.

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.
Note: If you have a sensor on the EMG that gives you other data than V then it is in the SI unit of that type of data, except for accelerometers that are in g.

Force
Struct array with data from the force plates. The elements of the struct array contain the data of the respective force plates present in the file.

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 meter (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.

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.

ForcePlateOrientation
Coordinate system in which force data is expressed: 0 (local force plate coordinates), 1 (global coordinate system).

RigidBodies
Struct with data for the 6DOF bodies. The struct contains the data of all rigid bodies present in the file.

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 "Appendix F: Rotation angle calculations in QTM".

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.

Filters
Struct with information about the used filter for the respective rigid bodies.

Preset
A list with the names of the used filter presets for the respective rigid bodies.

Skeletons
Struct array with data of the skeletons. The elements of the struct array contain the data of the respective skeletons present in the file.

SkeletonName
The name of the skeleton [char array].

NrOfSegments
Number of segments of the skeleton [double].

SegmentLabels
Names of the segments of the skeleton [1 x *NrOfSegments* cell array with char elements].

PositionData
Position data (X, Y, Z) of the segments of the skeleton [3 x *NrOfSegments* x *Frames* double].

RotationData
Segment rotation data in global coordinate system expressed as quaternions (QX, QY, QZ, QW) [4 x *NrOfSegments* x *Frames* double].

SMPTETimecode
Struct array with the SMPTE timestamps of the frames in the file.

Hour
The hour of the timestamp.

Minute
The minute of the timestamp.

Second
The second of the timestamp.

Frame
The SMPTE frame number of the timestamp.

Missing
Indicates if the SMPTE timecode is extrapolated if the SMPTE synchronization source is lost during the measurement.

Note: In QTM versions 2.15 and earlier an additional field **Subframe** is included, representing an index of marker frames within a SMPTE frame.

Note: The values are represented as int64 numbers in Matlab.

IRIGTimecode
Struct array with the IRIG timestamps of the frames in the file.

Year
The year of the timestamp.

Day
The day of the timestamp.

Hour
The hour of the timestamp.

Minute
The minute of the timestamp.

Second
The second of the timestamp.

Tenth
The decisecond of the timestamp.

Missing
Indicates if the timecode is extrapolated if the IRIG synchronization source is lost during the measurement.

Note: The values are represented as int64 numbers in Matlab.

CameraTimecode
Struct array with the camera timestamps of the frames in the file. For more information about Camera Time, see chapter “Timestamp” on page 162.

Tick
Tick value of camera timestamp. This value represents the time in seconds multiplied with a factor 10^7.

Missing
Indicates if the timecode is extrapolated if the synchronization source is lost during the measurement.

Note: The values are represented as int64 numbers in Matlab.

Events
Struct array with a list of all the events in the file.
Note: This struct array is only included if the capture file contains events.

Label
The label of the event.

Frame
The corresponding frame of the event. The time will be rounded to the nearest frame.

Time
The time of the event.

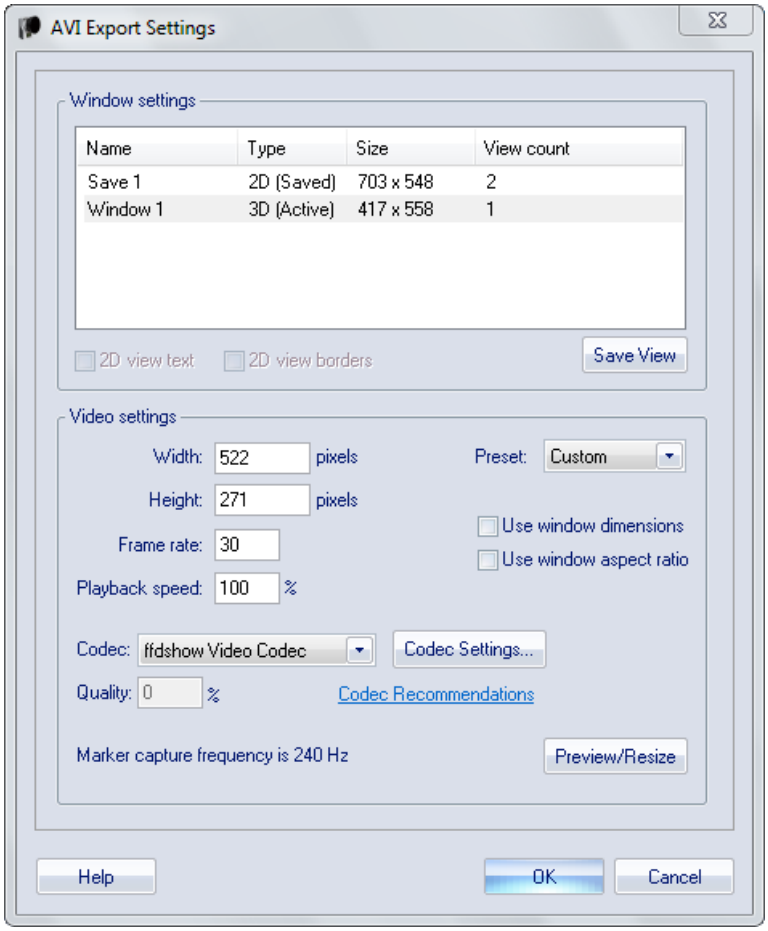
GazeVector and **EyeTracker**
The contents of these fields are described in the chapters “Process and export Dikablis gaze vector data” on page E - 9 and “Process and export SMI gaze vector data” on page E - 18.

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 to AVI file

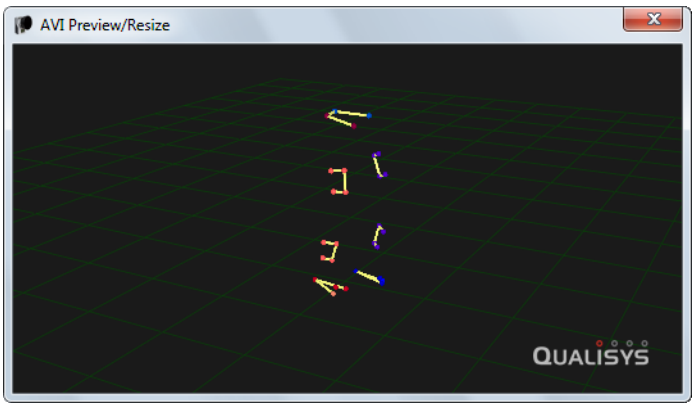
With the export to AVI file option you can create a video file of any camera view, 2D view or 3D view window. There are three different ways to make an export, the options for the exports are the same and are described in the chapter “AVI Export” on page 235.

File menu
On the File/Export/To AVI menu you can export to an AVI file from the currently open file. You can select any open window or a saved view to be used for the export. The settings that are used in the export are then copied to the **Project options**. This include the **Previous settings** view in the Window settings list.



A 3D view window will have the current rotation and zoom throughout the AVI file so it is important to set them as you want them. In a 2D view window the currently active camera views are included in the export, however when resizing the video dimensions the positions of the view may change compared to the original layout.

Use the **Preview/Resize** button to open a window where you can check the dimensions and also change the dimensions.



Right-click on 3D or camera view
On the right click menus of the 3D and 2D view windows there is an option to export that view to an AVI file. In this case only the current view is available for export and the settings are not copied to the **Project options**.

Note: Only the camera view that you right-click on is exported from the 2D view and the dimensions of the video will default to number of pixels used in the capture. If you want to export several camera views then you can use the export to AVI option on the **File** menu.

Processing step
The AVI export can be done as processing step either directly after a measurement, in reprocessing or in batch processing. The view that is used in the export is the one saved in **Previous settings** in the Window settings list. Since the **Previous settings** view is changed when you make an export, it is important that you save any view that you want to use with the **Save view** option. Then you can select that view again for the processing step before the processing.

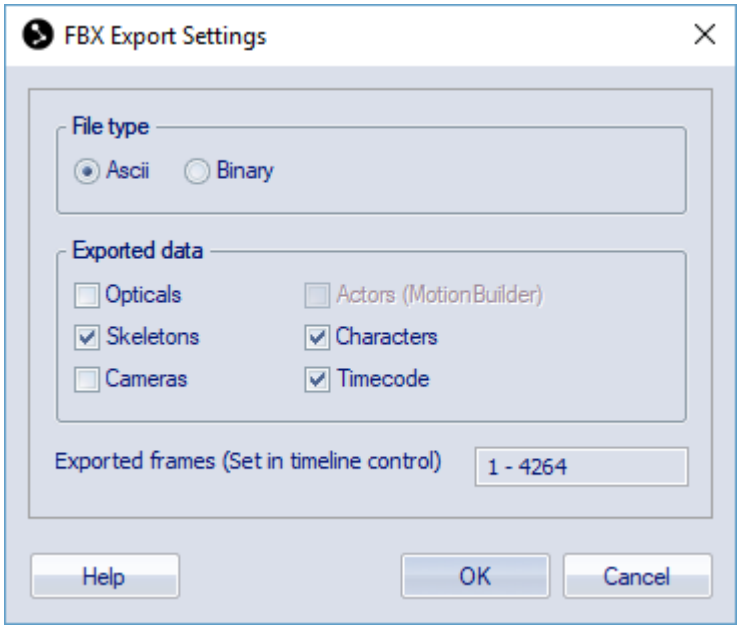
It is recommended to use a codec when creating the AVI file, since the video file will be very large uncompressed. The recommended codec is FFDShow/MJPEG. This codec is fast and can be played on most computers. However if you need a lossless codec, then you can use Lagarith. For more information about codecs see chapter “Codecs for Oqus high-speed video files” on page 291.

The video export is done by displaying the view that you want export in a special window that is then captured to an AVI file. Because the video has to be displayed in the window, the export will be faster on a computer with a good external graphics board. The processing time can also be reduced by using no more than 30 Hz and by making the dimensions of the video export smaller.

If you are exporting a Qualisys video to an AVI file, e.g. with a 3D overlay, then the video image will be linearized. I.e. the same parameters that are used to correct the marker data is applied to the video data to correct for the lens distortion. Therefore the pixels will not match exactly with the original video image. The linearization can be turned off with the **Show linearized data** option on the **2D view settings** page in **Project options**.

Export to FBX file

Click **Export** on the **File** menu and then **To FBX** to export to the FBX format. The FBX format is widely used for exchange of 3D data for animation software. The FBX format is especially useful for exporting skeleton-related data in QTM for the use in animation software such as MotionBuilder, Maya or Blender. For more information about the types of exported data, see chapter “FBX export” on page 238.



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 F - 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 206. By default QTM uses the **Qualisys standard** definition, which is described in the chapter “Rotation angles in QTM” on page 344.

System hardware

Qualisys cameras and devices

This chapter presents and overview of the available Qualisys devices and instructions on how to set up a Qualisys system. The topics covered are:

- Qualisys device types: an overview of Qualisys devices for a wide range of capture applications and environments.
- Description of Qualisys devices: a detailed description of the parts and connectors of the available Qualisys devices.
- Setting up the system: detailed instructions on how to set up camera systems and how to mix different camera types.
- Network configuration: detailed instructions on how to configure the computer for use with a Qualisys system.
- Firmware update: instructions on how to update the firmware of Qualisys devices.

Qualisys camera types

Qualisys cameras are available for a wide range of capture applications and environments. The following chapters give an overview of the available types and adaptations for special environments.

Marker cameras

Qualisys offers a variety of marker cameras for Arqus, Miqus and Oqus systems. For an overview and specifications of the various types and models, see “ Qualisys camera sensor specifications (marker mode)”.

The marker cameras calculate the positions and sizes of the detected markers on the camera, which then are sent to the computer over the camera network. This leads to a great reduction of information, which allows the cameras to be connected in a daisy chain. The edges of the markers are detected by using an intensity threshold. The center and size of the markers is calculated in subpixels, a subdivision of each pixel in 64 units per dimension.

The following features are available for Qualisys marker cameras:

Sensor mode
Most marker cameras have the possibility to use different sensor modes. For an overview of available sensor modes per camera model, see “ Qualisys camera sensor specifications (marker mode)”. The available sensor modes are:

High-resolution mode: in high-resolution mode the full resolution of the sensor is used. This is the default sensor mode.

High-speed mode: in high-speed mode one fourth of the pixels is used, allowing for increased capture rates at full field of view.

Marker size filtering
Size filtering can be used to filter markers that are smaller or larger than a set minimum or maximum threshold.

Marker masking
Marker masking can be used to block marker detection at selected areas of the sensor. For more information, see chapter “Marker masking” on page 297.

Circularity filtering
Filtering of markers based on their circularity level. This feature is supported by all Oqus marker cameras. For more information, see chapter “Marker circularity filtering (Oqus)” on page 299.

Exposure delay
Exposure delay can be used to improve marker detection in setups with opposing cameras by shifting the exposure of one or more cameras relative to other cameras in the system. For more information, see chapter “Delayed exposure to reduce reflections from other cameras” on page 259.

Active filtering
Active filtering can be used to improve marker detection in environments with high background illumination. Active filtering works by capturing each image twice to remove the background light. This dramatically increases the ability to capture passive markers in an outdoor environment. Active filtering is supported by Oqus 3+, 4, 5+, 6+ and 7+ series cameras. For more information, see chapter “How to use active filtering” on page 296.

Marker types
All marker cameras support the use of different types of markers, namely passive and several types of active markers. For more information, see chapter “IR markers” on page 261.

Video mode
All marker cameras can be used in video mode. This can be helpful, for example for pointing the cameras when setting up the system. Most cameras have the possibility to stream compressed video images. For an overview, see “ Qualisys video sensor specifications (in-camera MJPEG)”.

Video cameras

Qualisys offers several types of cameras for recording video. The main types of video cameras are:

- Several types of video cameras for streaming video, see chapter “Streaming video” on the facing page.
- A hybrid camera that can be used both as a marker camera and a color video camera, see chapter “Miqus Hybrid” on the facing page.
- High-speed video cameras for recording of buffered high-speed video sequences, see chapter “Oqus high-speed video camera” on page 412.

For more information on how to use Qualisys video cameras, see chapter “Qualisys video capture” on page 286.

Streaming video

Streaming video can be recorded with Miqus Video Color/Mono or the Oqus color video camera (Oqus 2c). When using streaming video, compressed video data is sent to QTM during the capture, allowing for long video captures.

For more information on capturing streaming video, see chapter “Capture streaming video” on page 287.

For more information about streaming video cameras, see the chapters below.

Miqus Video

The Miqus Video series is a dedicated video camera for capturing of MJPEG compressed video. There are two types: Miqus Video Mono (grayscale image) and Miqus Video Color. The Miqus Video series is configured with a white strobe and a built-in filter that filters out the IR light for a better image quality. The camera is always configured for video using In-camera MJPEG compression for effective streaming. The maximum video capture frequency at full resolution is 86 Hz. For information about max video capture frequency at different presets see chapter “Maximum capture rate for streaming video” on page 288.

It is possible to use up to three Miqus Video cameras in a single Qualisys system. For the use of more than three Miqus Video cameras a special setup is required, see chapter “Setting up Miqus Hybrid or Video for markerless mocap” on page 431.

For more information on how to use the Miqus Video camera see chapter “Miqus Video camera” on page 287.

Oqus color video camera (2c-series)

The Oqus color video camera (2c-series) is a dedicated video camera for capturing of MJPEG compressed video. It is configured with a clear glass to let in the visible light. There is a filter on the lens that filters out the IR light for better image and the strobe is white.

The camera is always configured for capturing streaming video (MJPEG) by default. When switching off the In-camera MJPEG compression, it can also be used as a high-speed video camera.

For more information on how to use the 2c-series camera see chapter “Oqus color video camera” on page 287.

Miqus Hybrid

The Miqus Hybrid camera is a two-in-one camera which can be used for both marker tracking and color video recording. The dual functionality of the camera makes it especially useful for the exploration of markerless applications.

In Marker mode, the camera uses the near IR strobe (850 nm) for the illumination of markers. The resolution and frame rate are similar to the Miqus M3 camera.

In Video mode, the specification are similar to the Miqus Color Video camera.

Note: The Miqus Hybrid camera is designed for indoor use. In Marker mode, the camera is more sensitive to ambient light compared with dedicated marker cameras. In Video mode colors may become washed out due to infrared light from the sun.

Note: It is discouraged to use the Miqus Hybrid cameras in Video mode together with other cameras in Marker mode. This will lead to washed out colors due to the near IR illumination from the other cameras.

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 - 13.

For more information on how to capture high-speed video, see chapter “Capture high-speed video” on page 289.

Weather protected cameras

Qualisys provides weather protected cameras that are specially adapted for outdoor use or use in industrial environments.

The housing, including connectors and cabling, is IP67/NEMA 6 classified, making the whole system water- and dust resistant. All Arqus and Oqus models are available in protected edition. Additional features for improving outdoor measurements are:

- The sun filter, a narrow band pass filter that effectively filters out the sun light while maximizing the visibility of the IR reflections of the markers. The sun filter is included with all Arqus protected cameras.
- Active filtering, the possibility to subtract background illumination from the reflected light of the markers.
- An extended operating temperature range of -15° to 45°C for Arqus protected cameras.
- The possibility to use long-range active markers.

Underwater systems

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 cameras. The camera housing is IP68 classified and pressure tested to 5 bar (40 m depth). All cameras are pressure tested to a depth of 40m and are corrosion-protected for use in salt water tanks or chlorinated swimming pools. Weight and volume is balanced to give the camera neutral buoyancy for easy handling in water.

There are several camera models available, currently:

- Miqus marker and video cameras with wide field of view,
- Oqus marker cameras with a powerful strobe for capturing at longer distances.

Qualisys underwater cameras are equipped with a special strobe with high power cyan LEDs. These LEDs are not limited to a flash time of 10% of period time as the regular strobe. 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.

The cameras are connected with underwater cables to a connector unit on land. One connector unit can be used to connect up to three cameras. For more than three cameras the respective connector boxes can be chained together or connected via an Ethernet switch.

Warning: When using an underwater camera above water the strobe unit can easily overheat. Be careful when testing the camera out of water and do not use exposure times longer than 1000 µs.

Note: The FOV is changed by refraction index of water (1.33). For example, a lens with 40º FOV is reduced to 31º FOV when used underwater.

MRI compatible cameras

Qualisys provides cameras, including cabling and accessories adapted for use inside MRI rooms. A minimal camera system of only three cameras can be sufficient to track hand and finger movements outside the scanner bore. A larger six camera system can provide the possibility to look into the bore, for example for monitoring the movements of the head during a scan.

The housing is made of die-casted aluminium, steel fasteners are replaced by brass screws and the calibration L-frame and wand is made from glass fiber. The camera and cables are completely shielded to keep electro magnetic emissions to a minimum and enabling cameras to operate just a meter away from the scanner without causing artifacts in the MRI image. The camera system is connected with the computer in the control room via optical Ethernet communication. The power supply is outside the room and connected via a filter through the MRI room panel.

The current line up consists of several Oqus camera models for capturing markers or color video with HD resolution.

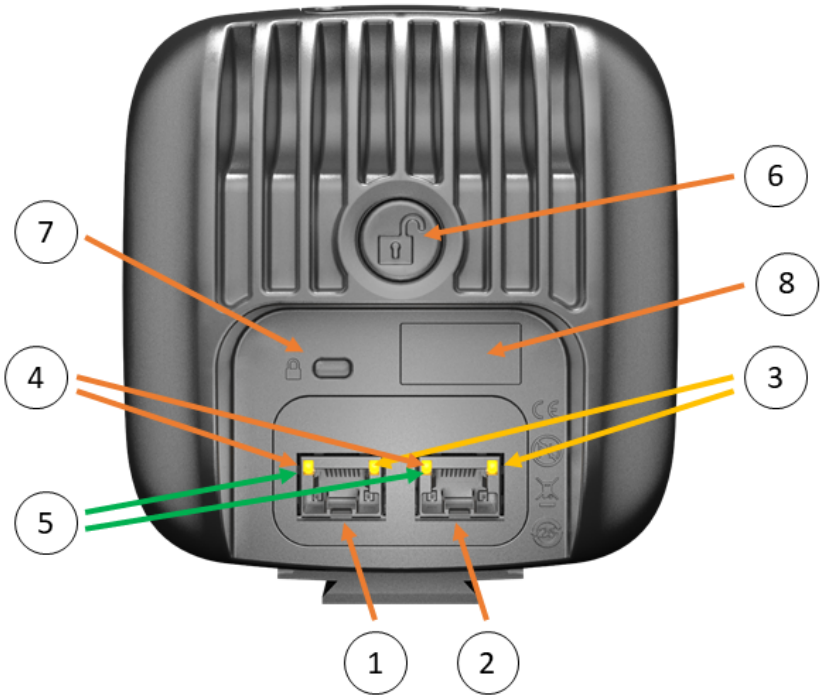
Description of Qualisys devices

Description of Arqus cameras

Arqus camera: front side

1. **LED ring**
   LED ring for camera identification and indication of status during startup sequence.
   - Green light: Active camera indicator (see “Identifying the cameras” on page 256).
   - Pulsing green light: Camera system being calibrated.
   - Amber light: Camera is booting.
   - Pulsing amber light: Camera is waiting for IP address.
2. **Measurement status indicator**
   - Green light: The system is ready to start a measurement.
   - Yellow light: The system is measuring.
   - Flashing green light: The camera is synchronizing to the master device.
   - Flashing red light: Error signal.
3. **Camera display**
   - Display of camera ID.
4. **Mounting plate**
   - Quick release for Manfrotto and Arca Swiss, two 1/4" camera mounts.

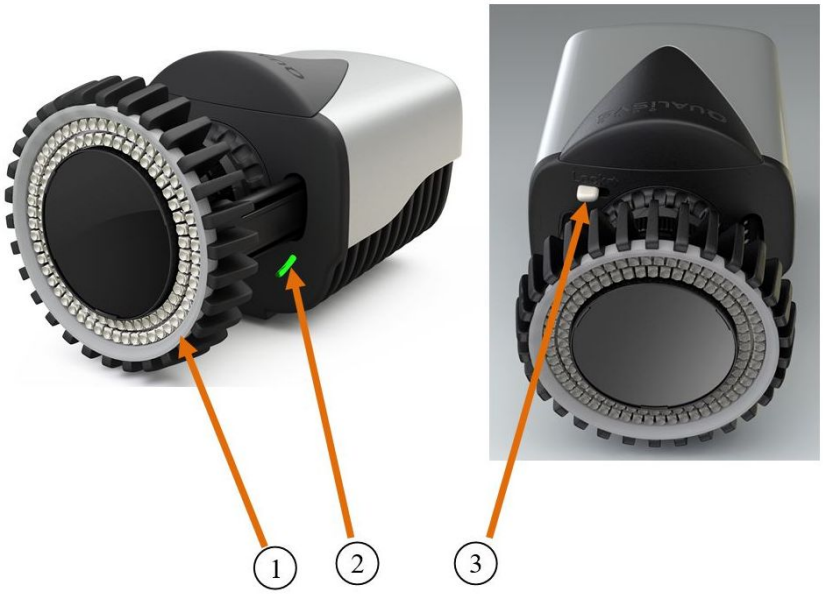
Arqus camera: back side



1. **Left daisy-chain data/power port**
   Combined Power/Gigabit Ethernet connector.
2. **Right daisy-chain data/power port**
   Combined Power/Gigabit Ethernet connector.
3. **Left/right port Link/Activity indicator**
   Shows the status of the ethernet connection. Fixed yellow light means that a carrier signal has been sensed and that the connection is up. Flashing yellow light indicates that data is received and/or transmitted.
4. **Left/right port Gigabit link indicator**
   Lit orange when Gigabit Ethernet link is established.
5. **Left/right port Power indicator**
   Lit green when the camera is powered on.
6. **Strobe unlock button**
   Press/release to unlock/lock the strobe mechanics.
7. **Kensington Security Slot**
   Slot for connecting a security cable.
8. **Camera identification**
   This label shows:
   - The serial number of the camera.
   - The product number.
   - The Ethernet Mac address.

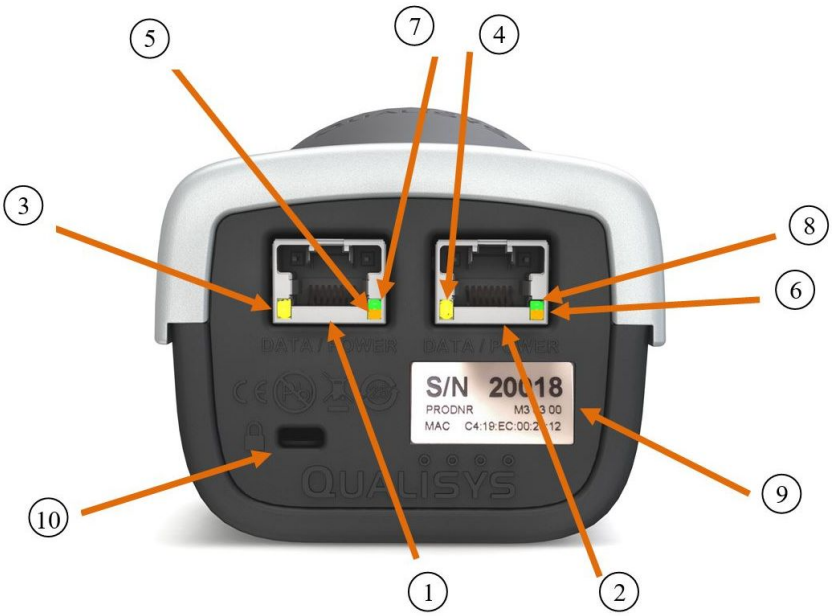
Description of Miqus cameras

Miqus camera: front side



1. **LED ring**
   LED ring for camera identification and indication of status during startup sequence.
   - Green light: Active camera indicator (see “Identifying the cameras” on page 256).
   - Pulsing green light: Camera system being calibrated.
   - Amber light: Camera is booting.
   - Pulsing amber light: Camera is waiting for IP address.
2. **Measurement status indicator**
   - Green light: The system is ready to start a measurement.
   - Yellow light: The system is measuring.
   - Flashing green light: The camera is synchronizing to the master device.
   - Flashing red light: Error signal.
3. **Lock lever**
   - Lock/unlock the strobe mechanics.

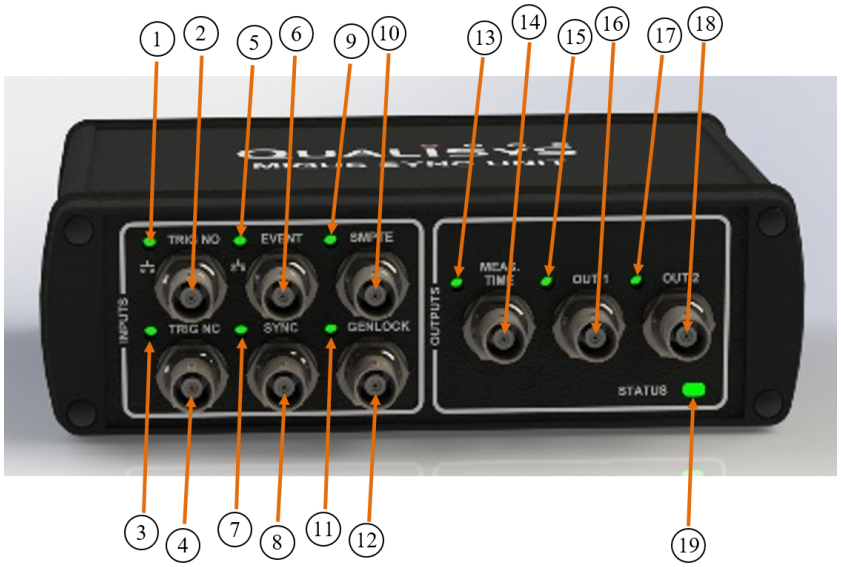
Miqus camera: back side



1. **Left daisy-chain data/power port**
   Combined Power/Gigabit Ethernet connector.
2. **Right daisy-chain data/power port**
   Combined Power/Gigabit Ethernet connector.
3. **Left port Link/Activity indicator**
   Shows the status of the ethernet connection. Fixed yellow light means that a carrier signal has been sensed and that the connection is up. Flashing yellow light indicates that data is received and/or transmitted.
4. **Right port Link/Activity indicator**
   Identical to left indicator.
5. **Left port Gigabit link indicator**
   Lit orange when Gigabit Ethernet link is established.
6. **Right port Gigabit link indicator**
   Identical to left indicator.
7. **Left port Power indicator**
   Lit green when the camera is powered on.
8. **Right port Power indicator**
   Lit green when the camera is powered on.
9. **Camera identification**
   This label shows:
   - The serial number of the camera.
   - The product number.
   - The Ethernet Mac address.
10. **Kensington Security Slot**
    Slot for connecting a security cable.

Description of Camera Sync Unit

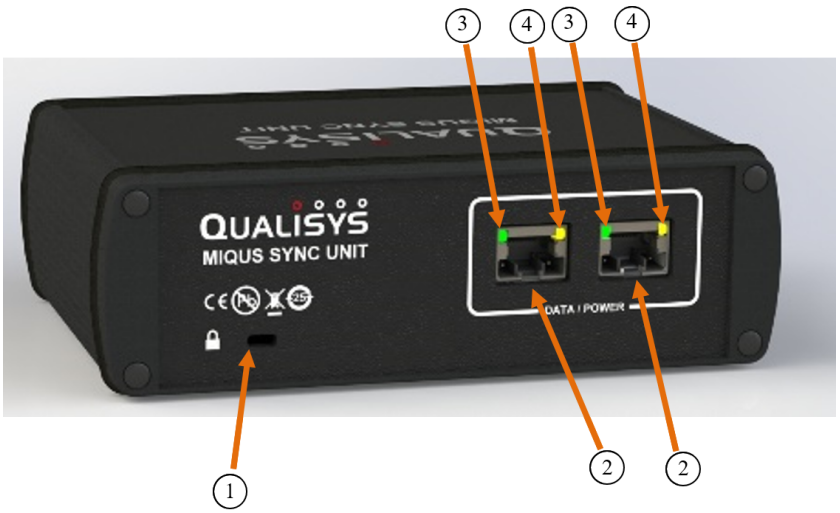
Camera Sync Unit: front side



1. **Trig NO indicator**
   Turns on for 0.5 s when the Trig NO input transitions from high to low.
2. **Trig NO input**
   Trigger input (TTL, 0-5 Volt, normally open). The base voltage of the port is 5 Volt (high).
3. **Trig NC indicator**
   Turns on for 0.5 s when the Trig NC input transitions from low to high.
4. **Trig NC input**
   Trigger input (TTL, 0-5 Volt, normally closed). The base voltage of the port is 0 Volt (low).
5. **Event indicator**
   Turns on for 0.5 s when the Event input transitions from high to low.
6. **Event input**
   Input for creating events in QTM. The base voltage of the port is 5 Volt (high).
7. **Sync indicator**
   Turns on for 0.5 s when the Sync input transitions from low to high.
8. **Sync input**
   Input for TTL synchronization signal (0-5 Volt). The base voltage of the port is 0 Volt (low).
9. **SMPTE indicator**
   Turns on when a valid SMPTE signal is present at the SMPTE input.
10. **SMPTE input**
    Input for SMPTE time code signal, e.g. from a MOTU sound device.
11. **Genlock indicator**
    Turns on 0.5 s for each vsync frame when a valid video signal is present at the Genlock input.

12. **Genlock input (video)**
    Genlock input for synchronization to a video signal. Compatible with Composite video (s-video) and Component video (YPbPr / GBR).
13. **Measurement time indicator**
    Turns on 0.5 s at the start of a measurement.
14. **Measurement time output**
    Outputs a pulse that lasts for the duration of the measurement.
15. **Output 1 indicator**
    Turns on for 0.5 s when the Output 1 port transitions from low to high.
16. **Output 1**
    Programmable synchronization output (TTL, 0-5 Volt).
17. **Output 2 indicator**
    Turns on for 0.5 s when the Output 2 port transitions from low to high.
18. **Output 2**
    Programmable synchronization output (TTL, 0-5 Volt).
19. **Measurement status indicator**
    - Green light: The system is ready to start a measurement.
    - Yellow light: The system is measuring.
    - Flashing green light: The device is synchronizing to the master device.
    - Flashing red light: Error signal.

Camera Sync Unit: back side



1. **Kensington Security Slot**
   Slot for connecting a security cable.
2. **Daisy-chain data/power port**
   Combined Power/Gigabit Ethernet connector.
3. **Power/Gigabit Link indicator**
   Lit green when the Sync unit is powered on. An additional orange LED is lit when a Gigabit link is established.
4. **Link/Activity indicator**
   Shows the status of the Ethernet connection. Fixed yellow light means that a

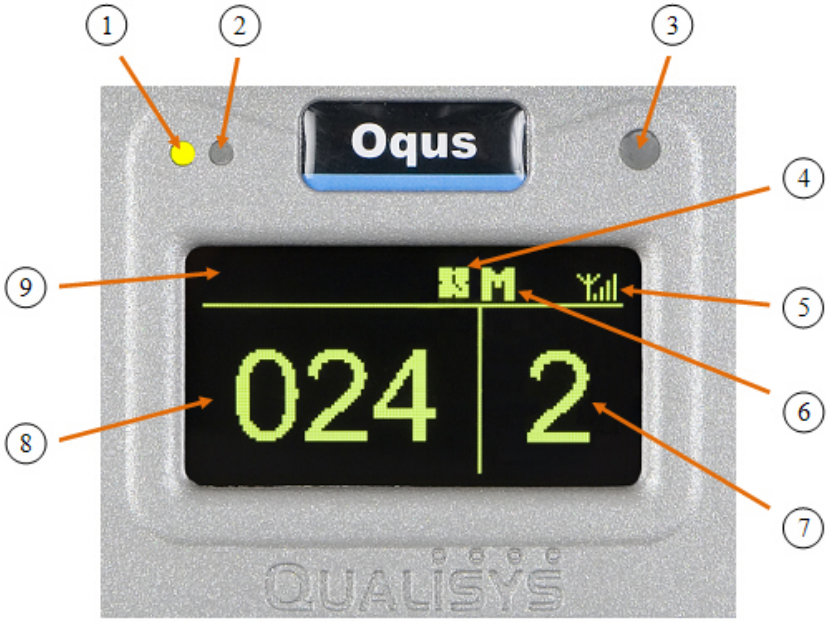
carrier signal has been sensed and that the connection is up. Flashing yellow light indicates that data is received and/or transmitted.

Description of Oqus devices

Oqus camera 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 437.
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 camera 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.

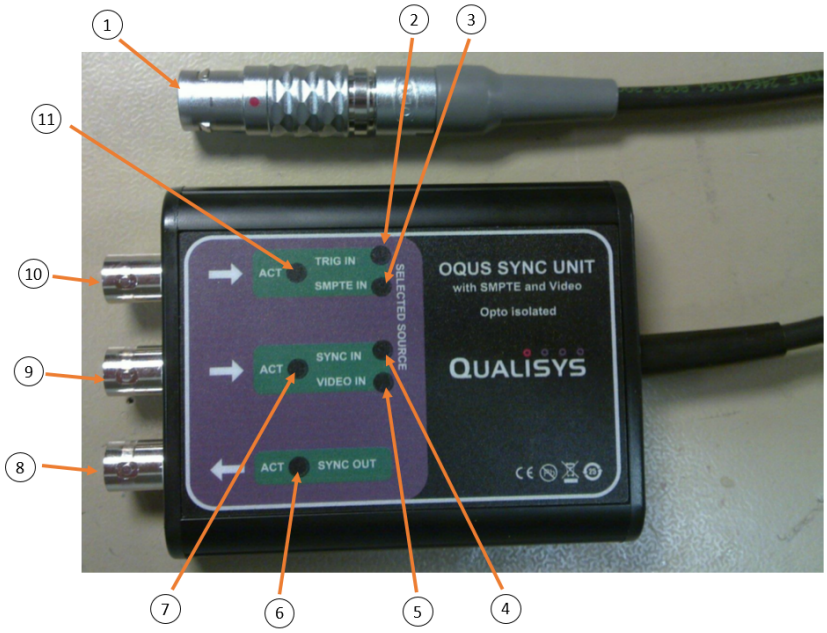
1. **Left Data port** (light blue)
   Ethernet connector. 100BaseTX/802.3i, 100 Mbps, Fast Ethernet.
2. **Right Data port** (light blue)
   Identical to the left data port.
3. **Left Ethernet activity indicator**
   Shows the status of the Ethernet connection. Fixed green light means that a

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 - 15.
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.

Oqus Sync Unit



1. **Oqus connector**
   Connector for control port on an Oqus camera.
2. **Trig In indicator**
   Lit when in Trig in mode.
3. **SMPTE indicator**
   Lit when in SMPTE mode.
4. **Sync In indicator**
   Lit when in Sync in mode.
5. **Video In indicator**
   Lit when in Video/Genlock mode.
6. **Sync out activity indicator**
   Lit green for 0.5 s when output transitions from high to low.
7. **Sync in activity indicator**
   Lit green for 0.5 s when signal transitions from high to low.
8. **Sync output**
   Programmable synchronization output (TTL, 0-5 Volt).
9. **Sync input/Genlock (video) input**
   Mode dependent on **Signal source** selected in QTM (see chapter “External timebase” on page 159).
   - Sync input (TTL, 0-5 Volt) when **Signal source** set to **Control port**.
   - Video input when **Signal source** set to **Video sync**. Compatible with Composite video (s-video) and Component video (YPbPr / GBR).
10. **Trig input**
    Mode dependent on QTM project settings.

- Trigger input (TTL, 0-5 Volt) when in Trig in mode (default).
- SMPTE timecode input (e.g. from a MOTU sound device) when in SMPTE mode (**Signal source** set to **SMPTE** or **Use SMPTE timestamp** activated, see chapters "External timebase" on page 159 and "Timestamp" on page 162).

11. **Trig in activity indicator**
    Lit green for 0.5 s when signal transitions from high to low, or continuously lit when a valid SMPTE signal is present.

Setting up the system

There are three main Qualisys system platforms, Arqus, Miqus and Oqus, each with several types of cameras or other devices. All devices are compatible and can be connected and combined in many ways. The following chapters provide guidelines for how to set up different system configurations.

Before connecting a Qualisys camera system, make sure that the QDS (Qualisys DHCP server) is running on the computer and that the network interface settings are correct, see "QDS" on page 437 respectively "Network card setup" on page 437.

For further information about optimizing the camera setup and settings, see chapter "Optimizing the camera system and settings" on page 254.

Camera system setup: technical notes

It is possible to combine Qualisys devices in a system in many ways. However, there are some technical limitations related to the type of cables used, the power requirements of the cameras and the bandwidth. The technical aspects of connecting different types of cameras are summarized below.

Inter-chain mixing
Cameras using the same cable type can be mixed together in a chain, without the need of a network switch. To mix cameras of different cable types you need to use a switch. For an overview of cable types, see the Table below.

Power kit capacity
Due to varying power consumption, the number of cameras a power kit can power depends on the camera type. For an overview of the maximum number of cameras per power kit, see the Table below.

Maximum chain sizes in homogeneous systems
The number of cameras that can be connected in a single daisy chain is limited. The maximum chain size per camera type is indicated in the Table below. To build a camera system with a larger number of cameras you need to use multiple chains, which are joined together with a network switch.

Camera type	Cable type	Max. power kit capacity	Max. chain size
Arqus	A	5	20
Arqus Protected	B	5	20
Miqus	A	10	20
Miqus Video	A	3	3

Camera type	Cable type	Max. power kit capacity	Max. chain size
Miqus Hybrid	A	3	3
Oqus	C	5	15

Note: You can add a **Camera Sync Unit** to the chain beyond the maximum power kit capacity or maximum chain size.

Mixing Arqus and Miqus

Arqus and Miqus cameras can be mixed in the same chain. Due to differences in power consumption, one Arqus camera can be interchanged by two Miqus cameras and vice versa. Supported combinations per power kit:

- 4x Arqus + 2x Miqus
- 3x Arqus + 4x Miqus
- 2x Arqus + 6x Miqus
- 1x Arqus + 8x Miqus

Adding Miqus Hybrid or Miqus Video cameras

Power consumption wise, a Miqus marker camera can be swapped out for a Miqus Hybrid or Miqus Video camera. However, due to bandwidth requirements it is recommended to limit the number of video cameras in a chain to a maximum of two video cameras.

If more video cameras are needed, they should be added in a separate chain and joined through a network switch.

Important: If you need more than three video cameras, the faster 10 Gigabit switch is required.

Setting up a Qualisys system with an Ethernet switch

For most systems the recommended setup is a daisy chain configuration. However, Qualisys systems can also be connected through an Ethernet switch. The use of a switch allows for more flexibility in setting up and combining Qualisys devices. The switch should have a bandwidth of at least 1 Gigabit. For systems with more than three Miqus video cameras a 10 Gigabit switch is required.

To setup the system with an Ethernet switch, follow these instructions:

1. Check that the computer has an Ethernet card with at least the same bandwidth as the switch.
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 page 437.
4. Connect the respective daisy chains of Qualisys devices to one port each.

   Note: Do not connect the different daisy chains to each other. It will disrupt the communication and the switch will be of no use.

The use of a switch is required in the following cases:

Large camera systems
When the number of cameras in a system exceeds 20 for Arqus and Miqus or 15 for Oqus, the daisy chains should be split up in smaller chains.

More than two Qualisys video cameras in a system
When there are more than two video cameras in a Qualisys system a switch is needed for optimizing the data communication. For Oqus high-speed cameras, the fetching time can be greatly reduced by connecting a maximum of 2-3 cameras per port.

Systems with more than three Miqus Video or Miqus Hybrid cameras
This type of setup requires a 10 Gigabit switch with a maximum of three cameras per port.

Not all Ethernet switches are suitable for the use with a Qualisys system. Please contact Qualisys AB for specific recommendations.

Setting up an Arqus system

An Arqus system consists of the following components:

Arqus cameras
Arqus cameras of one or several types.

Camera cables
The number of camera cables is the same as the number of cameras.

One or more Power kits
A power kit consists of a power supply, a power injector and a host (Ethernet) cable. One power kit can power up to five Arqus cameras.

Camera Sync Unit (optional)
A Camera Sync Unit is required for synchronization with external equipment.

Gigabit Ethernet switch (optional)
The use of a switch is required for systems with more than 20 Arqus cameras.

The Arqus system is easy to set up. The backside of each camera contains two **Data**/**Power** ports. In a basic setup with up to 5 cameras, the cameras (and, optionally, a Camera Sync Unit) are connected to each other by Camera cables in a daisy chain configuration (see illustration below). The first camera or Camera Sync Unit is connected via a **Power injector** to the power supply and the computer as follows:

1. Connect the power supply to the **Power** port on the power injector.
2. Connect one end of the host cable to the to the **Data** port on the power injector and the other end to the **Ethernet** port of the computer.
3. Connect one end of the Camera cable to the **Camera** port on the power injector and the other end to a **Power/Data** port of the first camera or the Camera Sync Unit.

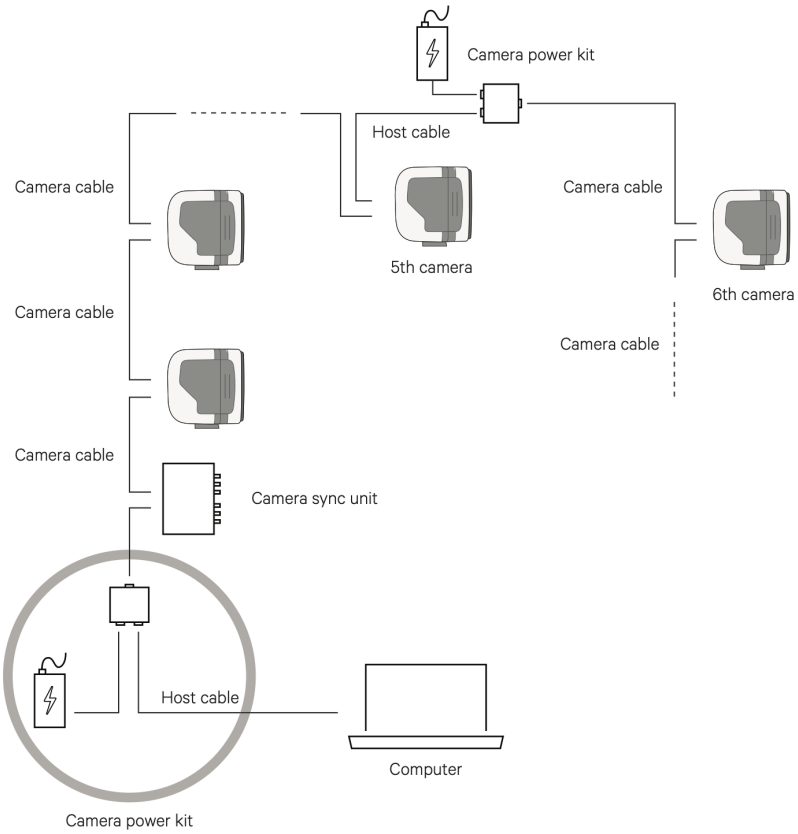
For larger systems, start a new power chain when the maximum of 5 cameras per power chain is reached. Connect the first camera of the new power chain (camera 6, 11, 16) via a **Power injector** as follows:

1. Connect the power supply to the **Power** port on the power injector.
2. Connect one end of the host cable to the to the **Data** port on the power injector and the other end to a **Power/Data** connector of the previous camera.

3. Connect one end of the Camera cable to the **Camera** port on the power injector and the other end to a **Power/Data** connector of the first camera in the next power chain.

For systems with more than 20 cameras, the use of a switch is required. The system can then be subdivided in chains with up to 20 cameras. The chains are connected with their respective host cables to a switch, which in its turn is connected to the computer.

The **Camera Sync Unit** can be placed anywhere in the chain, but in most cases it will be practical to have it close to the computer.



When the cables have been connected correctly, the indicator LEDs at the Arqus data/power ports will indicate the status of the power and data connection. For more information, see chapter “Arqus camera: back side” on page 415.

The Arqus startup sequence

Before you connect the Arqus camera system, make sure that the QDS (Qualisys DHCP server) is running and that the network interface settings are correct. This is needed for the cameras to receive an IP address from QDS to communicate with other Miqus cameras and the host computer. For more information, see “QDS” on page 437 and “Network card setup” on page 437.

The general Arqus startup sequence is as follows (total duration about 50 s).

1. Connect the power to the cameras.
2. Booting of the cameras. The amber LED ring is lit during booting.
3. The camera receives an IP address.
   If the camera does not receive an IP address the amber LED ring pulses. Possible reasons for this might be that the system is not connected to the

computer, or that QDS is not running. For instructions on how search for the error, see Section“Troubleshooting connection” on page 491.

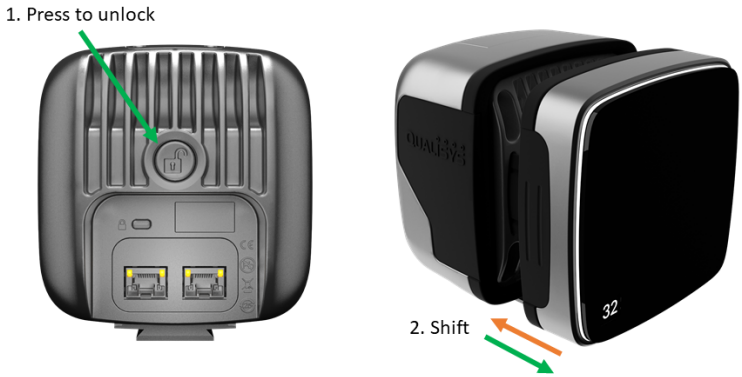
4. The cameras synchronize to the master camera. The green status LED on the front of the camera blinks during synchronization. When synchronization is achieved it lights solid green.

Setting aperture and focus

The Arqus A12 with standard lens option has a motorized lens, which can be controlled from QTM via the **Lens Control** interface in the **Camera Settings** sidebar in the 2D View window, see “Camera settings sidebar” on page 31.

For Arqus cameras with a manual lens, you need to extend the strobe mechanics to get access to the focus and aperture rings of the lens. The strobe mechanics can be shifted as follows:

1. Press and hold the **Strobe unlock** button at the backside of the Arqus camera.
2. Shift the strobe mechanics outwards to expose the lens for adjustment.
3. Shift the strobe mechanics inwards again when done. You may have to fine adjust the position of the strobe mechanics so that it locks into the dimples on the strobe rails.



The strobe should always be in closed position during measurement to achieve the best strobe light distribution.

For general recommendations on settings aperture and focus, see chapter “Tips on setting aperture and focus” on page 257.

Setting up a Miqus system

A Miqus system consists of the following components:

Miqus cameras
Miqus cameras of one or several types.

Camera cables
The number of camera cables is the same as the number of cameras.

One or more Power kits
A power kit consists of a power supply, a power injector and a host (Ethernet) cable. One power kit can power up to 10 Miqus cameras.

Camera Sync Unit (optional)
A Camera Sync Unit is required for synchronization with external equipment.

Gigabit Ethernet switch (optional)
The use of a switch is required for systems with more than 20 Miqus cameras.

The Miqus system is easy to set up. The backside of the camera contains two **Data/Power** ports. In a basic setup with up to 10 cameras, the cameras (and, optionally, a Camera Sync Unit) are connected to each other by Camera cables in a daisy chain configuration (see illustration below). The first camera or Camera Sync Unit is connected via a **Power injector** to the power supply and the computer as follows:

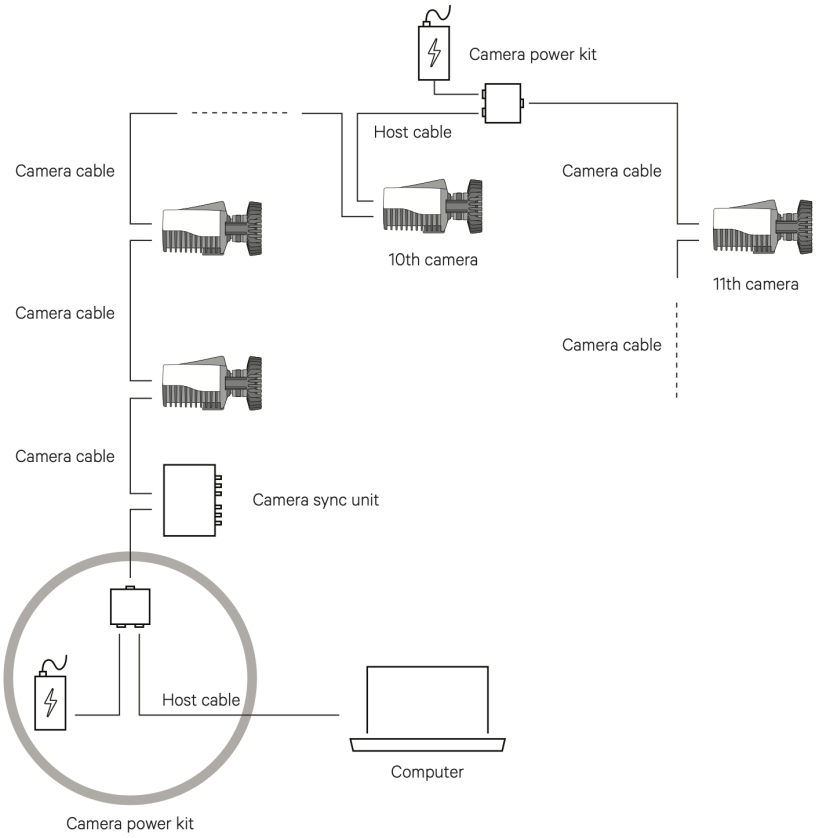
1. Connect the power supply to the **Power** port on the power injector.
2. Connect one end of the host cable to the to the **Data** port on the power injector and the other end to the **Ethernet** port of the computer.
3. Connect one end of the Camera cable to the **Camera** port on the power injector and the other end to a **Power/Data** port of the first camera or the Camera Sync Unit.

For larger systems, start a new power chain when the maximum of 10 cameras per power chain is reached. Connect the first camera of the new power chain (camera 11) via a **Power injector** as follows:

1. Connect the power supply to the **Power** port on the power injector.
2. Connect one end of the host cable to the to the **Data** port on the power injector and the other end to a **Power/Data** connector of the previous camera.
3. Connect one end of the Camera cable to the **Camera** port on the power injector and the other end to a **Power/Data** connector of the first camera in the next power chain.

For systems with more than 20 cameras, the use of a switch is required. The system can then be subdivided in chains with up to 20 cameras. The chains are connected with their respective host cables to a switch, which in its turn is connected to the computer.

The **Camera Sync Unit** can be placed anywhere in the chain, but in most cases it will be practical to have it close to the computer.



When the cables have been connected correctly, the indicator LEDs at the Miqus data/power ports will indicate the status of the power and data connection. For more information, see chapter “Miqus camera: back side” on page 417.

The Miqus startup sequence

Before you connect the Miqus camera system, make sure that the QDS (Qualisys DHCP server) is running and that the network interface settings are correct. This is needed for the cameras to receive an IP address from QDS to communicate with other Miqus cameras and the host computer. For more information, see “QDS” on page 437 and “Network card setup” on page 437.

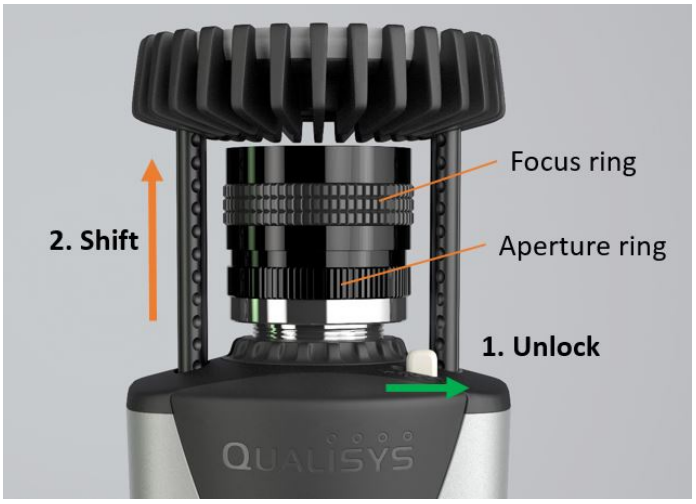
The general Miqus startup sequence is as follows (total duration about 50 s).

1. Connect the power to the cameras.
2. Booting of the cameras. The amber LED ring is lit during booting.
3. The camera receives an IP address.
   If the camera does not receive an IP address the amber LED ring pulses. Possible reasons for this might be that the system is not connected to the computer, or that QDS is not running. For instructions on how search for the error, see Section“Troubleshooting connection” on page 491.
4. The cameras synchronize to the master camera. The green status LED on the front of the camera blinks during synchronization. When synchronization is achieved it lights solid green.

Setting aperture and focus

All Miqus cameras are equipped with a manual lens. You need to extend the strobe mechanics to get access to the focus and aperture rings of the lens. The strobe mechanics can be shifted as follows:

1. Unlock the strobe mechanics with the **Lock lever**.
2. Shift the strobe mechanics outwards to expose the lens for adjustment.
3. Shift the strobe mechanics inwards again when done and lock it with the lock lever. You may have to fine adjust the position of the strobe so that the lock lever locks into the dimples on the strobe rails.



The strobe should always be in closed position during measurement to achieve the best strobe light distribution.

For general recommendations on settings aperture and focus, see chapter “Tips on setting aperture and focus” on page 257.

Setting up Miqus Hybrid or Video for markerless mocap

Miqus Hybrid or Miqus video cameras can be used in systems for markerless mocap. A markerless mocap system consists of the following components:

Miqus Hybrid or Miqus Video cameras
Miqus Hybrid or Miqus Video cameras or a combination of both.

Camera cables
The number of camera cables is the same as the number of cameras.

One or more Power kits
A power kit consists of a power supply, a power injector and a host (Ethernet) cable. One power kit can power up to 10 Miqus cameras. However, since the maximum number of Miqus Hybrid or Miqus Video cameras per chain is limited to three, it is more practical to use one power supply per chain of three cameras.

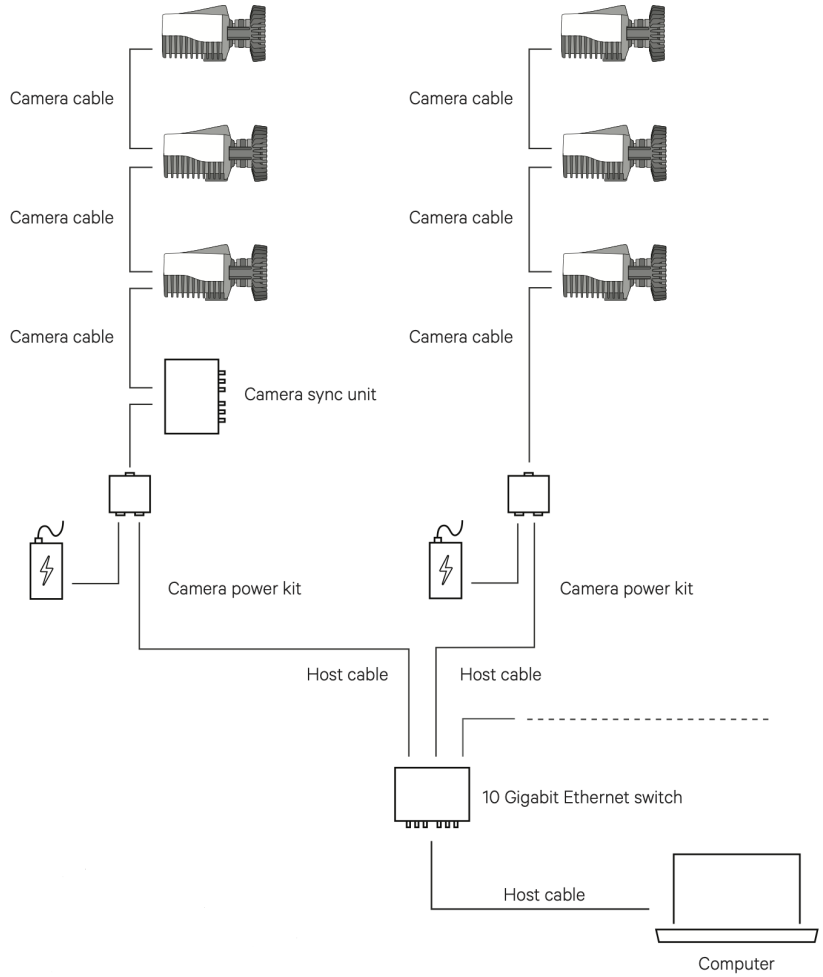
Camera Sync Unit (optional)
A Camera Sync Unit is required for synchronization with external equipment.

10 Gigabit Ethernet switch
The use of a 10 Gigabit switch is required for systems with more than three Miqus Video or Miqus Hybrid cameras.

Desktop Markerless Computer
A specially configured computer is needed with a 10 Gigabit network adapter and a disk configuration optimized for handling large amounts of video data.

A system with Miqus Hybrid or Miqus Video is set up in the same way as a Miqus system. However, the maximum chain size is limited to three cameras. The chains must be connected via a 10 Gigabit switch to the Desktop Markerless Computer.

Note: A maximum number of 16 Miqus Hybrid or Miqus Video cameras can be combined in a markerless system.



Setting up an Oqus system

An Oqus system consists of the following components:

Oqus cameras
Oqus cameras of one or several types.

Hybrid Power/Data cables
Cables carrying both power and data between cameras.

Data cables
Cables carrying data between cameras.

Host cable
Cable carrying data between first camera in the chain and computer or switch.

One or more power supplies
One power supply can power up to five Oqus cameras.

Trigger/Sync splitter (optional)
A Trigger/Sync splitter cable can be connected to the **Control** port of an Oqus camera for synchronization with external devices. It is also possible to add a **Camera Sync Unit** to the system. However, this requires the use of an Ethernet switch.

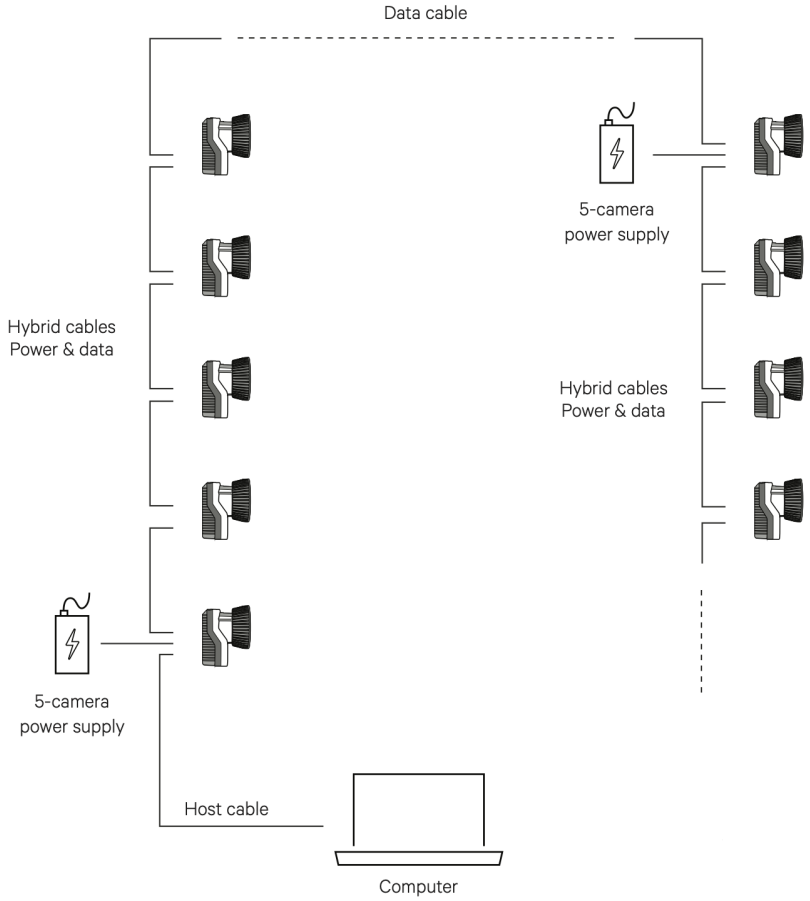
Gigabit Ethernet switch (optional)
The use of a Gigabit Ethernet switch is required for systems with more than 15

Oqus cameras or if you have more than two Oqus 2c or Oqus high-speed video cameras.

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 camera connectors” on page 421.

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.

For larger systems, start a new power chain when the maximum of 5 cameras per power chain is reached. One of the **POWER** ports of the first camera of the new chain should be connected to a power supply. The **DATA** ports of the last camera of the previous chain and the first camera of the new chain are connected with a **Data cable**.



Note: For Oqus systems larger than 15 cameras and for systems with many high-speed cameras, the performance can sometimes be improved with a Gigabit Ethernet switch and then connect the cameras in shorter daisy-chains, see “Setting up a Qualisys system with an Ethernet switch” on page 425.

Oqus startup sequence

Before you connect the camera system, make sure that the QDS (Qualisys DHCP server) is running and that the network interface settings are correct. This is needed for the cameras to receive an IP address from QDS to communicate with other Miqus cameras

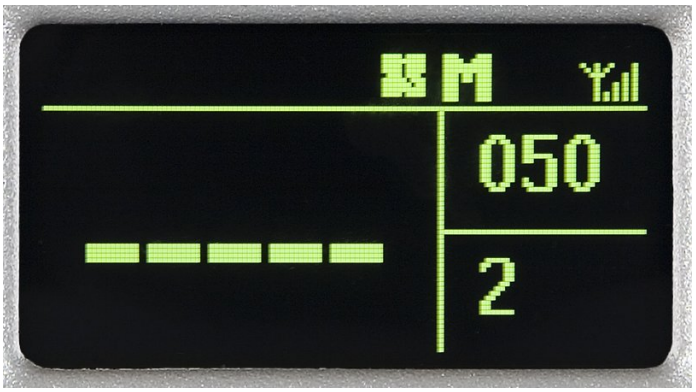
and the host computer. For more information, see “QDS” on page 437 and “Network card setup” on page 437.

The general Oqus startup sequence is as follows.

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 491.

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

The Oqus 7+ with standard lens option has a motorized lens, which can be controlled from QTM via the **Lens Control** interface in the **Camera Settings** sidebar in the 2D View window, see “Camera settings sidebar” on page 31.

For Oqus cameras with a manual lens, you need to extend the strobe mechanics to get access to the focus and aperture rings of the lens.

1. Turn the strobe mechanics counterclockwise to expose the lens for adjustment.

2. When done, turn the strobe back (clockwise) to close it.

The strobe should always be in closed position during measurement to achieve the best strobe light distribution.

For general recommendations on settings aperture and focus, see chapter “Tips on setting aperture and focus” on page 257.

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.

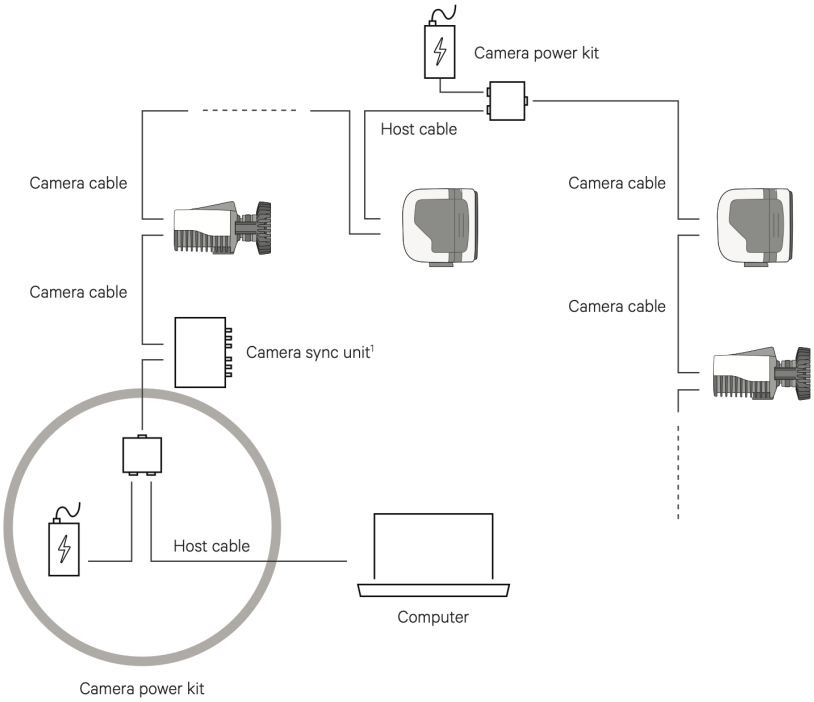
Setting up Oqus for wireless communication requires that the wireless adapter of the computer is set up as a hosted network. This requires a computer running Windows 7. This feature is no longer supported for most wireless adapters in Windows 10. The configuration requires a special version of QDS, contact support@qualisys.com for more information.

Setting up a mixed system

All Qualisys devices are compatible and can be combined in a single system.

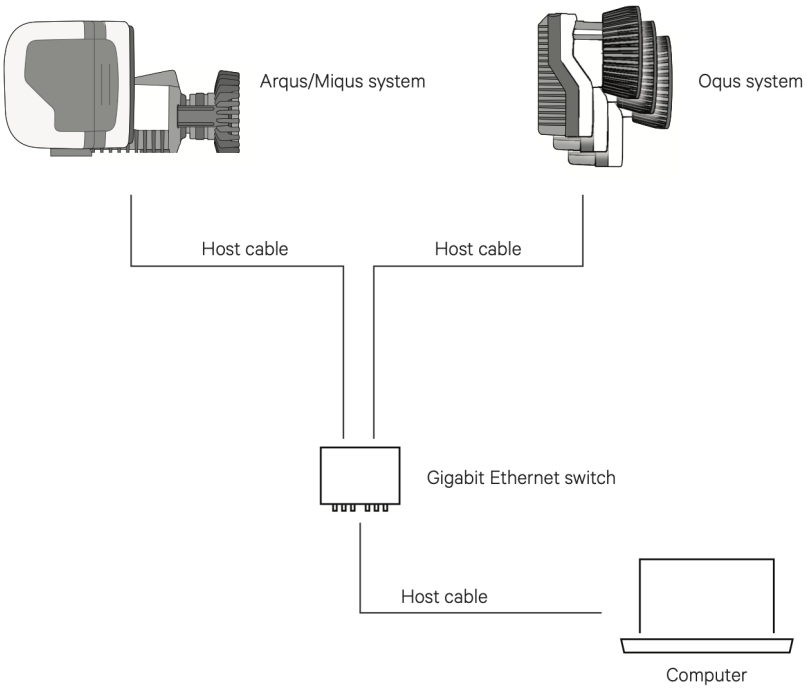
When mixing camera types the global camera settings will default to the lowest of the camera limits. This means for example that the capture rate will be limited to 183 Hz at full field of view when Miqus M5 is combined with Miqus M3. However, individual settings can always be set within the limit of the camera type.

Devices with the same cable type can be connected in a daisy chain, for example Arqus and Miqus cameras with cables of type A, see “Camera system setup: technical notes” on page 424. The total number of cameras that can be connected to a power supply depends on the number of Arqus and Miqus cameras, see chapter “Mixing Arqus and Miqus” on page 425.



For combining devices with different cable type the use of a Gigabit Ethernet switch is required. For example, for mixing Arqus or Miqus cameras with Oqus cameras, the respective systems are connected in their own daisy chain configurations. The chains are then connected with host cables to the Ethernet switch, see chapter “Setting up a Qualisys system with an Ethernet switch” on page 425 for more information.

Not all Ethernet switches are suitable for the use with a Qualisys system. Please contact Qualisys AB for specific recommendations.



Network configuration

Network card setup

It is recommended to use a computer with two network interfaces, one reserved for the Qualisys camera 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 Qualisys camera system.

IMPORTANT: Do not connect the Qualisys camera 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 Qualisys camera network, other setups are possible but not recommended by Qualisys. Use the wizard to setup the network interface, see “Network configuration wizard” on page 439.

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 Qualisys camera 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., for example 192.168.254.1. 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.
7. Then enter the **Subnet mask** as 255.255.255.128. Click OK twice.

From QTM 2019.3 wireless connection of an Oqus system is no longer supported by the included version of QDS. Please contact Qualisys support at support@qualisys.com for a special version of QDS if you need to connect an Oqus wireless camera to a wireless network.

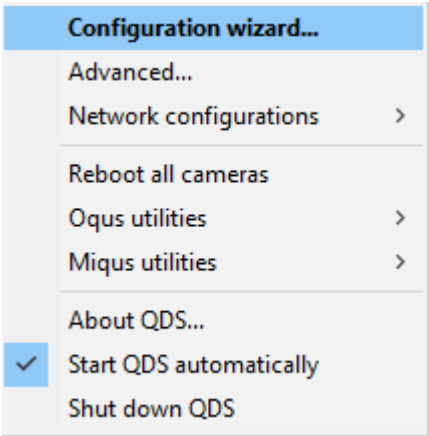
QDS

QTM comes with a DHCP server called QDS (Qualisys DHCP Server), which distributes IP addresses to the Qualisys 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 Qualisys cameras so it will not disturb your computer network.

QDS includes a wizard to configure network interfaces to use them with Qualisys cameras, see “Network configuration wizard” on page 439.

QDS menu

To open the QDS menu, right-click on the QDS icon in the status toolbar.



The QDS menu contains the following options:

Configuration wizard
The configuration wizard can be used to setup your network interface for Qualisys cameras, to configure an Oqus camera for wireless operation and to setup a virtual wireless access point, see “Network configuration wizard” on the facing page.

Advanced
Advanced configuration of network interfaces on the computer, see “Advanced (network settings)” on page 441.

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**.

Reboot all cameras
Reboot all Qualisys cameras that have an IP-address.

Oqus utilities
QDS can control the Oqus cameras with the following commands.

Display on
This is the default mode of the camera. The display when you use the camera, but goes into sleep mode if not used for 2 hours.

Display off
In this mode the camera display and front LED is off. They do not turn on unless you enable them with **Display on** or reboot the cameras.

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.

Miqus utilities
QDS can control the Miqus cameras with the following commands.

Green on
Switch on green LEDs on the LED ring.

Green off
Switch off green LEDs on the LED ring.

Green pulsing
Pulse green LEDs on the LED ring.

Amber on
Switch on amber LEDs on the LED ring.

Amber off
Switch off amber LEDs on the LED ring.

Amber pulsing
Pulse amber LEDs on the LED ring.

About QDS
Information about QDS.

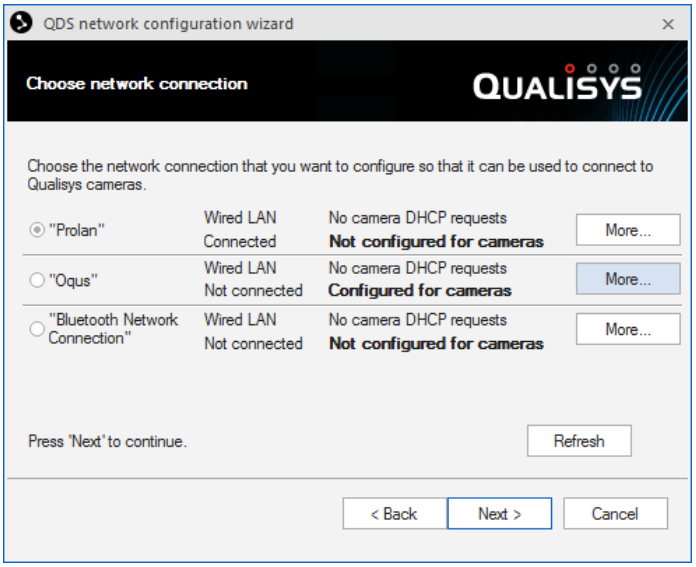
Start QDS automatically
Option for starting QDS automatically on computer startup.

Shut down QDS
Shut down QDS.

Network configuration wizard

The QDS network configuration wizard will guide you through the different steps to setup the network for Qualisys 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 437. Follow the steps below.

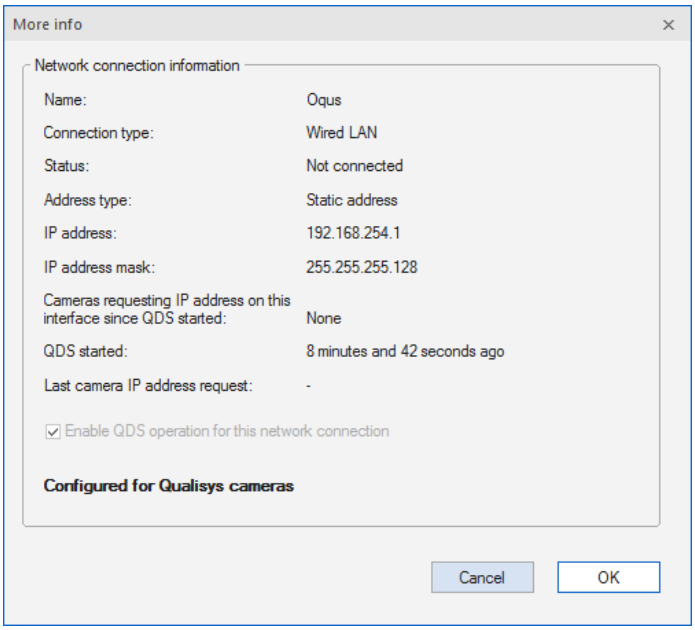
1. Double click on the QDS icon in the Windows task bar or click on **Configuration wizard** in the QDS menu to start the wizard.
2. Select **Configure a network interface...** and click **Next**.



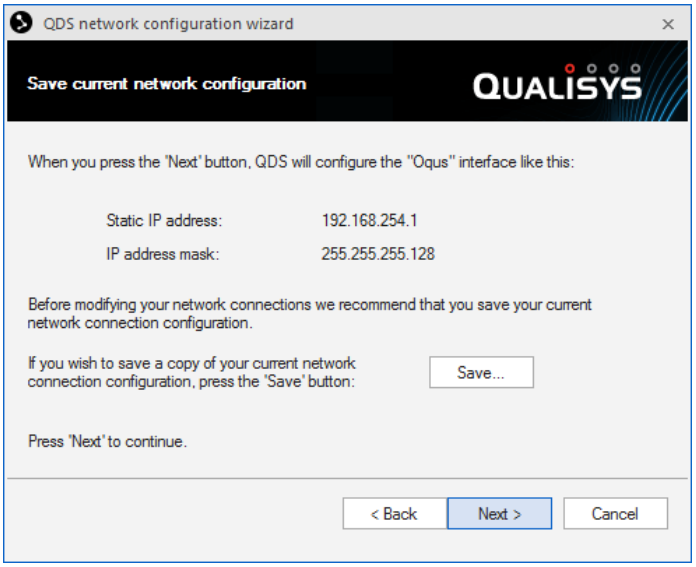
3. The list will show all enabled network interfaces on the computer. Select the interface that you want to use with Qualisys cameras and click **Next**.

   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.

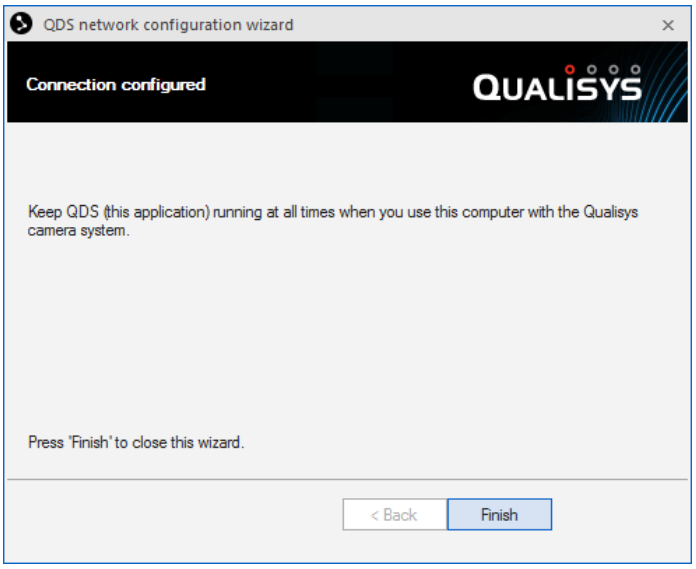
a. Click on **More** to see information about the network interface.



The "More info" window shows the current settings of the network interface. This is the same information as is shown in Advanced, see “Advanced (network settings)” on the facing page.

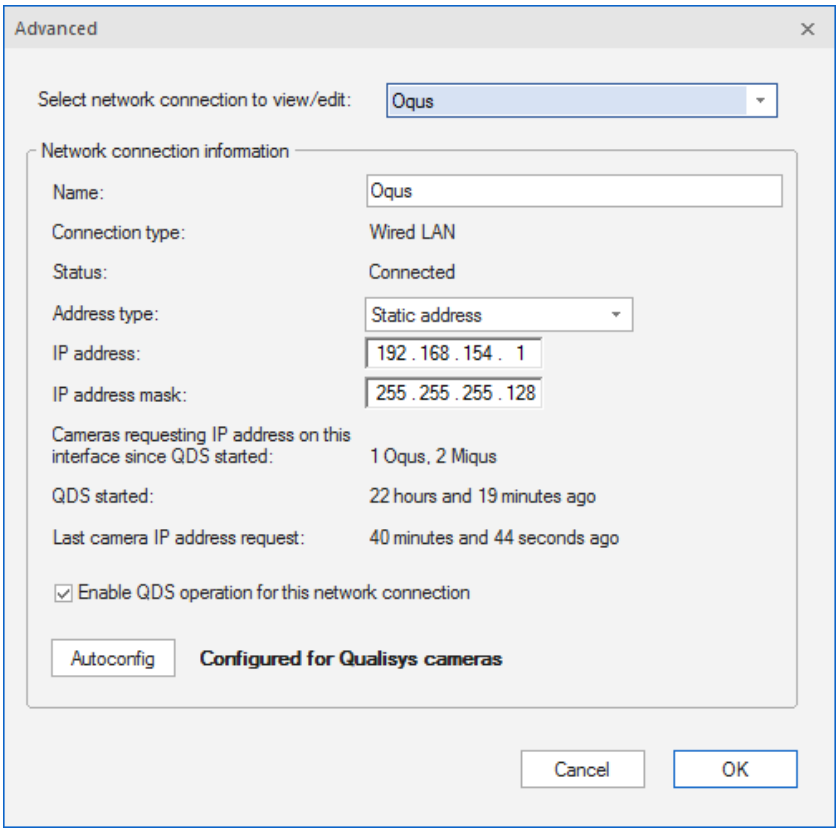


4. The wizard shows how it will change the selected interface. You can save the current network setup with the **Save** button for backup. Click **Next** to continue.



5. Click **Finish** to close the wizard.

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 Qualisys 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**.

Cameras requesting IP address on this interface since QDS started
Number of Qualisys cameras requesting IP address on the network.

QDS started
Time since QDS started.

Last camera IP address request
Time since last camera IP address request.

Enable QDS operation for this network connection
Activate QDS on the network, i.e. if QDS is disabled for the network, Qualisys 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.

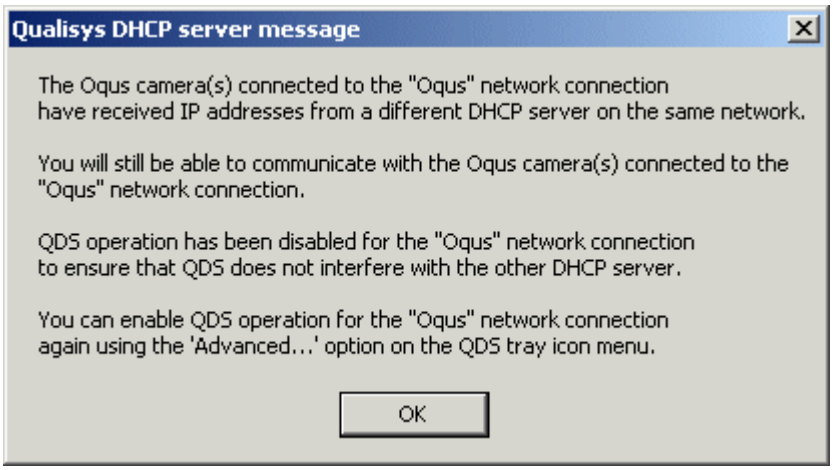
All of the networks with static IP addresses have been disabled.

Autoconfig
Use this button to configure the network interface for Qualisys 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 camera system otherwise QDS operation will be turned off again at the camera startup.



Firmware update

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 **Camera System** 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 **Camera System** 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 “Appendix B: Qualisys Firmware Installer (QFI)”.

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 "Appendix B: Qualisys Firmware Installer (QFI)".

Note: If the current camera firmware is a higher major version than QTM is adapted to the firmware will have to be downgraded instead.

Short range active marker

QTM can use both passive and active markers with the same camera system. The short range active marker is a light weight active marker, which can be used in situations when it is hard to use the passive markers. For example if there are a lot of unwanted reflections in the cameras. The short range active marker also have sequential coding which makes identifying the markers very simple, see chapter “When to use short range active marker” on page 449.

The active marker consists of a small driver, an IR eye and up to 32 active markers. It can only be used with the Oqus camera. For more information about the hardware see chapter “Short range active marker hardware” below.

Short range active marker hardware

The active marker consists of a driver unit capable of driving up to 32 markers through four daisy-chained outputs. The markers are small and light-weight and can be attached to the skin with simple double adhesive tape. The driver has an integrated battery which can either be charged in the driver unit or replaced with another battery and charged in a standalone charger. The battery is selected to last a day of normal measurements.

The active marker hardware consists of the following parts:

- A driver for up to 32 markers, see chapter “Short range driver” below.
- Selectable number of active markers on different chain lengths, and up to 4 marker chains with a maximum of 8 markers per chain, see chapter “Active markers and IR eye” on page 447.
- IR eye
- 2 batteries with power supply and an external battery charger, see chapter “Battery and charging” on page 447.
- Belt clip for driver
- Sync cable [Not included in the standard kit]

Short range driver

The driver is the center of the short range active marker. It has to be attached in a secure way to the subject to make sure that there is no strain on the cables. The active marker kit includes a belt clip that can be attached on the driver to fasten it for example on a belt.

The driver controls and powers the active markers and has the following connections and indicators.

Markers 1-32
4 connectors for the active marker chains. There can be up to 8 active markers on each chain and therefore there can be up to 32 markers on one driver, see chapter "Active markers and IR eye" on the facing page.

IR in/Sync In
Connector for the IR eye or Sync in. The standard use is to connect an IR eye to the input, The eye is triggered by a modulated pulse from the Oqus camera.
The sync in signal must be coded to be correct and must therefore be sent either from another active marker driver or from an Oqus camera with the sync out signal set to **Wired synchronization of active markers**.

Sync out
Use the Sync out connector to daisy chain several active marker drivers to one IR eye. Connect the Sync out signal from one driver to the Sync in connector on the next driver.

Charge/ 9 VDC
Use this connection to charge the battery with the supplied power supply. The driver can still be used while it is being charged.

WARNING: Do not use the driver with the power connected while the driver is mounted on a human being.

ON/OFF
Turn on the driver by pressing the button. Turn off by holding down the button a couple of seconds.

Driver ID
Change the driver ID to specify which sequential range to use for the driver. The possible ranges are 1-5, which corresponds to the IDs 1-32 to 129-160. 0 means that the sequential ID is turned off.

Battery indicator
When pressing the power button, the four LEDs indicates the battery status. One LED means that you soon have to charge the battery.

Status indicators
The three status indicators at the top displays the following information:

Power/Batt Low
The LED is lit green when the power is on and starts flashing green when the driver goes into the power save mode. The battery is low when the LED is lit or flashing red.

Charging
The LED is lit green when the driver is being charged, if something is wrong the LED is lit red.

Sync active
The sync is active when the LED is flashing orange.

Active markers and IR eye

The active markers are mounted on a daisy-chained wire that can have up to 8 markers and a length of 3 meters. The ID of the markers is decided by the connector on the driver. When a chain has less than 8 markers the other IDs of that connector will be skipped, e.g. if you have chain of 4 markers connected to the first two connectors these will have ID 1-4 and ID 9-12.

The IR eye is mounted on a short wire and can be mounted anywhere to detect the pulse from the camera. It has an adaptable sensitivity so it can easily detect the pulse in almost any setup, even outdoors.

Battery and charging

The battery in the short range active marker is a rechargeable Li-Polymer battery. It can power the active marker for a day of normal measurements. The battery can be recharged either in the driver with the power supply or separately in the battery charger. It takes about 3 hours to fully charge the battery.

WARNING: Do not use the driver with the power connected while the driver is mounted on a human being.

To remove the battery, press the button on the side of the driver and lift the bottom lid. Make sure that you insert the battery according to the image inside the driver.

QTM settings for the active marker

To activate the use of active markers you just select either **Active** or **Passive and Active** on the **Cameras** page. Then the cameras will start to send the pulsed signal which triggers the active markers. The two modes have the following differences:

Active
Only active markers are visible, which means that all other reflections from the camera flash are removed. This is because the cameras only send out a pulsed flash before the exposure.

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 lose some of the advantages of both types, see chapter “Passive and Active mode” below.

If you do not want to use the sequential identity codes, set the **Driver id** selector on the driver to 0 and then **Disable identity codes** on the **Cameras** page.

How to use sequential coding

The active markers can be used with sequential coding, this means that the marker identity is known automatically without an AIM model. QTM can handle up to 160 different IDs in a measurement. The IDs of the markers are controlled by the switch on the driver unit and it also depends on the connector that the marker chain is attached to. For example driver ID = 2 and the second marker connection will give you marker IDs 81-96.

The identification needs at least 21 frames to be able to identify the marker ID, any trajectory shorter than that will have ID 0. For the best result it is, therefore, often recommended to use cameras on two levels so that as many angles as possible are covered in the measurement volume.

Trajectory	ID #	Fill Le...	Range
Thigh_1	1	99.7%	4 - 1000
Thigh_2	2	99.6%	5 - 1000
Thigh_3	3	99.5%	6 - 1000
Thigh_4	4	99.4%	7 - 1000
Shank_1	5	100.0%	1 - 1000
Shank_2	6	100.0%	1 - 1000
Shank_3	7	100.0%	1 - 1000
Shank_4	8	99.9%	2 - 1000

The IDs of the markers are displayed in the **ID#** column of the **Trajectory info** windows. QTM will automatically join all of the trajectories with the same ID. However you can manually join any trajectories with an ID with a trajectory with ID 0, by dragging the one with ID to the one without ID. Then the joined trajectory will get the ID of the trajectory with trajectory. You can undo the merge with the **Undo** command.

To automatically identify the trajectories you need to make an AIM model. However, in this case you do not need any movement or correct placement of the markers to create the AIM model. You just capture a file with the correct marker IDs and then label them as you want to place them on the subject. Create the AIM model and the next time you apply the AIM model the trajectories will be labeled from their marker ID.

Note: In the **Passive and Active** mode the AIM model does not work the same way as when you only have active markers, see chapter “Passive and Active mode” below.

Passive and Active mode

The **Passive and Active** mode can for example be used in setups where you need to add some temporary markers for a static model. In that case it is often easier to add passive markers than to plug-in more active markers. There is however disadvantages to the mixed mode and it is therefore not recommended to be used in all of your

measurements. The disadvantages compared to just using the active markers are the following.

- The maximum frequency is reduced to 250 Hz because you need to flash the strobe both to activate the active marker and to see the passive markers. This also means that you will have less exposure time for the markers for example at 250 Hz the maximum exposure time is 210 µs.
- There can be unwanted reflections in shiny materials, because you have a flash of light during the exposure for the passive.
- If the AIM model includes both passive and active markers it has to be created from a file with the correct marker placement.

When to use short range active marker

There are benefits and downsides with the active marker compared to passive markers. The passive marker has two great benefits compared to the active marker and that is first of all that it is easier to place because there are no wires. Secondly it is more visible because it is more or less a sphere.

However the active markers also have some benefits. First of all there are no problems with other reflections and then it also has sequential coding which makes the identification very simple. Therefore the active markers are recommended if your setup is one of the following:

A lot of unwanted reflections
If there are a lot of unwanted reflections caused by a highly reflective environment, then the active marker is the best solution. Because then you will not have extra markers caused by the camera flash.

Simple marker identification
The sequential coding makes the identification very simple. The AIM functionality for passive markers can be just as effective as sequential coding, but in some setups the active markers can be easier to use. For example irregular movements of single markers cannot be handled by AIM and then the active marker is the best solution.

Outdoors
Because of the high intensity of the active marker it can be easier to use outdoors than passive markers. It can also be used at frequency up to 500 Hz, when the passive marker can only be used up to 250 Hz because of the active filtering.

Short range active marker specifications

Driver	83 x 52 x 15 mm, 99 g
IR marker	D 16 x H 11 mm, 3 g
Frequency range	1 - 500 Hz
Number of markers per driver	Up to 32 markers
Total number of markers	Up to 5 drivers can be used, up to 160 sequential coded markers
Maximum measurement distance	more than 25 m
Battery time, 200 fps, 16 markers	4,5 h
Time to fully charge battery	2-3 h

How to use analog boards

Connection of analog board

An analog board is needed on the measurement computer to capture analog signals together with the motion capture data. For information on how to install an analog boards see “Appendix C: Analog input/output hardware”.

The analog board is activated on the **Input Devices** page in the **Project options** dialog, see chapter “Input Devices” on page 121. The analog board settings are saved in the project with the serial number of the board. So after an analog board has been used in a project then it is visible even if you disconnect the board from the computer or it is not active in Instacal.

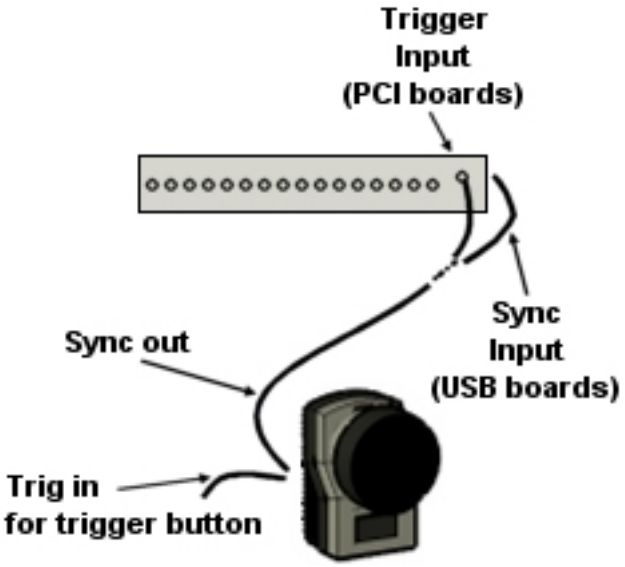
It is important to activate channels, where a cable is connected, on the page for the active analog board in the **Project options** dialog. It is also a good idea to not use the channels where nothing is connected, since these channels will only make the file larger.

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 “Sample rate” on page 167. The number of analog frames, therefore, does not have to be the same as the number of frames in the marker 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” on the facing page.

To get synchronized analog data and motion capture data, a synchronization signal must be sent to the analog board, either from a sync out connector on an Oqus camera or one of the sync out ports of the Camera Sync Unit.

Note: For Oqus it is recommended to connect the sync cable to the Master camera if the synchronization signal is faster than 5000 Hz.



USB-2533/USB-1608G

The recommended synchronization option is the frame synchronization. Activate

it with the **External sync (frame sync)** option for the analog board, see chapter "Sample rate" on page 167. For this option you must connect the **Sync out** connection from an Oqus camera or a Camera Sync Unit to the **Sync** connection on the back of the box.

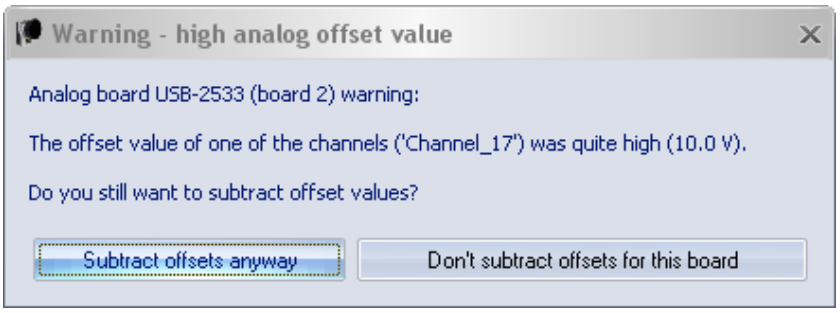
For USB-2533 the trigger edge that is used by the board can be set in Instacal see chapter "Installing the USB-2533 board" on page C - 3. Make sure that the **XAPCR edge** setting for the board matches the **TTL signal polarity** setting on the **Synchronization** page in the **Project options** dialog.

The other option is to start the analog capture with the sync output from a camera or sync unit. Then the frequency is set internally by the analog board, which means that there may be a small drift between the systems. To use this option connect the **Sync out** connection from an Oqus camera or one of the Outputs of the Camera Sync Unit to the **External trigger** or **Trig** connection on the A/D board. Then select the **Simultaneous start** option for the analog board, see chapter "Sample rate" on page 167.

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 168. 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.

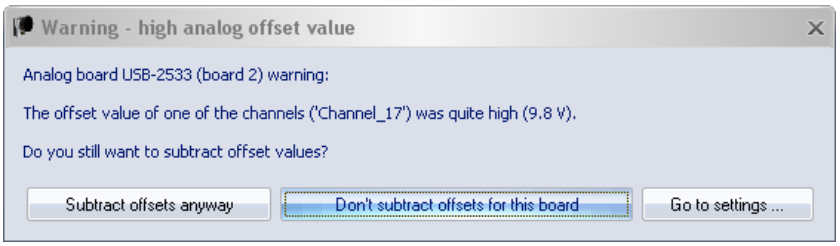
Note: If you have selected the **Remove offset in real-time** option, the warning will appear directly when you start the capture. The capture will however continue and if you like you can wait to choose what you want to do until the capture is finished. If you choose what to do before the file has finished, 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 91.

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 **Remove offset in real-time** 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.

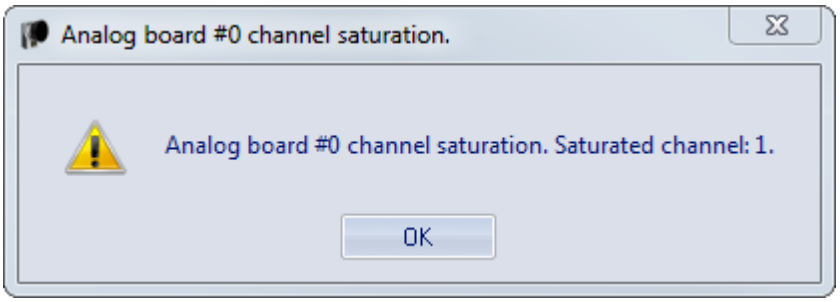
Subtract offset anyway
Subtract the analog offsets on the board in RT and keep the offset compensation options activated on the **Analog board** page.

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 170.

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 168.

Analog saturation warning



The analog data is checked after a measurement so that no channels have been saturated, i.e. that the analog value has not reached the limits of the range. This warning is shown in the status bar and in the **Messages** window.

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 next page and “How to use EMG” on page 463.

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 450.

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.

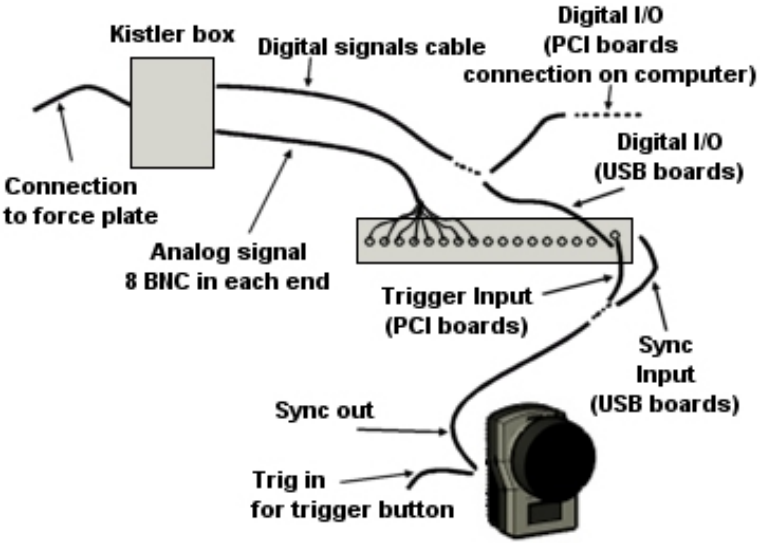
Note: The force plate must have an analog output with an output voltage between ± 10 V. However only the force plates listed in the chapter “Supported force plate devices” on page 462 can be used if you want to calculate the force in QTM.

Note: The channels must be activated on the page for the analog board in the **Project options** dialog, see chapter “Channels” on page 170.

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 with an analog board. 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. For information on the Kistler 5695A DAQ, see chapter “Connecting Kistler 5695A DAQ” on page 457.

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 5233A2.
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.

5233A2
You can either use a cable that controls two force plates per I/O port (230137) or a cable that controls one force plate on the first I/O port (230129).
Connect the DSUB15 end of the cable to port A on the analog board. Connect the DSUB37 end of the cable to the DSUB37 connector on the 5233A2 units.
Remember to press the Remote on the 5233A2 units to activate the digital control.
Note: If you want to use port B on the analog you must have a cable for 2 force plates in port A so that you can control the 3rd and 4th force plates from port B.

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

Note: The auto-zeroing of the force data can be disabled on the **Kistler Force-Plate Control Settings** page in the **Project Options** dialog, see chapter “Force plate auto-zero (Kistler)” on page 174.

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 C - 8. 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 settings** 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.

External Sync (recommended for Oqus)
Select the correct camera and the sample rate to get the desired frequency, see chapter "Sample rate" on page 167.

Trigger start
Specify the frequency 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 and add the number of Kistler force plates that you want to control to the list.

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, therefore you only need to add one force plate to the list.

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 213. They are found in the manual for the force plate. Use the option **Select by forceplate control**, so that the ranges used in the calculation are always correct.

b. Select the correct analog channels for each force-plate, see chapter "Kistler force plate settings" on page 216.

4. Enter the position of the force-plate, see chapter "Force plate location" on page 224. 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 5233A2 control unit.

Note: It is possible to see the current ranges on the 5233A2

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.

Connecting Kistler 5695A DAQ

The Kistler 5695A DAQ contains a USB-2533 analog board which is the same as the one used in the Qualisys 64 channel USB board. The DAQ can handle up to 8 Kistler force plates.

Note: QTM does not support the Kistler 5691A DAQ, because it cannot be frame synchronized and the Instacal driver can malfunction so that the DAQ disappears from the computer.

The description below will explain the connections that are specific for use with QTM. For information on how to connect the force plates, please refer to the Kistler manual.

1. Connect the USB connection to the computer and start Instacal.
   a. Make sure that you change the number of channels to 64 single-ended.
   b. When using the External sync option, make sure that that the **XAPCR Edge** setting matches the setting for the camera sync output on the **Synchronization** page in the **Project options** dialog. The default **TTL signal polarity** is **Negative** which matches **XAPCR Edge** = **Falling**.

   For more information about how to use Instacal see chapter “Installing the USB-2533 board” on page C - 3.
2. Connect the force plates starting with the first connector.

   The operate signal is sent automatically to reset the force plate before a measurement. It is, therefore, 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
3. Connect the Sync out signal from the cameras or the Camera Sync Unit to the Sync/trigger splitter [230133] on the Control I/O on the 5695A. The Kistler 5767 adapter box [230131] is also an alternative for the sync connection. Which input to use depends on the settings for the analog board:
   a. **External sync option** (recommended)
      Connect **Sync out** from an Oqus camera or the Camera Sync Unit to **Sync in** on the Kistler Sync/trigger splitter.
   b. **Not frame synchronized**
      Connect **Sync out** from an Oqus camera or the Camera Sync Unit to **Trig in** on the Kistler Sync/trigger splitter.

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 the analog board's settings page under **Analog boards** in the **Project options** dialog, see chapter “Analog board settings” on page 166. Make sure that it is the Kistler 5695A DAQ, because before you change the settings

below it will still be displayed as USB-2533.

a. Set the **Sample rate** for the analog board.

External Sync (recommended)
Select the correct camera and the sample rate to get the desired frequency, see chapter “Sample rate” on page 167.'

Trigger start
Specify the frequency 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.

b. Go the the **Force plate control settings** page in the **Project options** dialog.

i. Select the **Kistler 5695A DAQ** option.
IMPORTANT: If the option is not available make sure that you have installed the Kistler DAQ driver in the QTM installer.

ii. Add the force plates to the list and set the correct ranges, see chapter “Force plate control list” on page 174.

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 213. They are found in the manual for the force plate. Use the option **Select by forceplate control**, so that the ranges used in the calculation are always correct.

b. Select the correct analog channels for each force-plate, see chapter “Kistler force plate settings” on page 216.

4. Capture the position of the force-plate, see chapter “Force plate location” on page 224. 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.

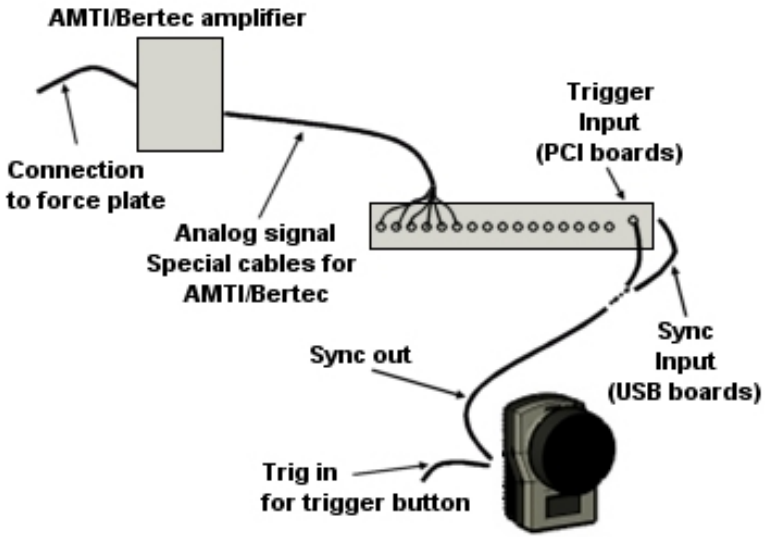
b. If there are no analog signals, start a measurement in BioWare and check if you can see the force data.

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.

Note: For the AMTI portable plate you need an extra box [art. no. 230009] from Qualisys ABthat has an output of the 8 analog channels as BNC.

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 from the AMTI (6 BNC cables) or AMTI portable and Bertec amplifier (8 BNC cables) to 6 respectively 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 C - 8. 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 settings** 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 211 and "Bertec force plate calibration parameters" on page 216.
      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 212 and “Bertec force plate settings” on page 217.

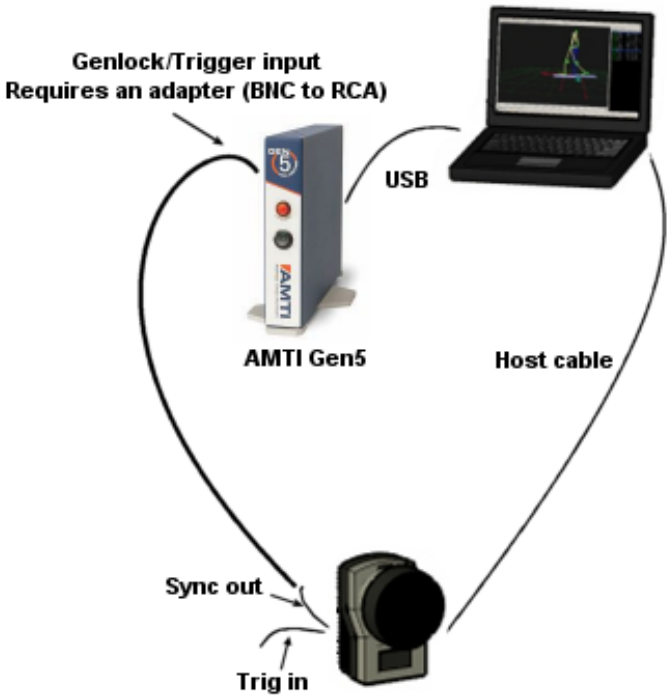
4. Enter the position of the force-plate, see chapter “Force plate location” on page 224. It is good to do this after each new calibration especially if the calibration L is not placed at the same position.
5. Activate the **Calculate force data** option on the **Processing** page. To see the force both 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.

Connecting AMTI Gen5/OPT-SC/AccuGait Optimized

When using an AMTI Gen5, OPT-SC (Optima Signal Conditioner) amplifier or the AccuGait Optimized plate you need to follow these steps. The steps are the same for AMTI Gen5, OPT-SC and AccuGait Optimized so the instructions only refer to Gen5.

1. Download the drivers to the computer. There is a **Download device drivers** link on the **Input Devices** page in **Project Options** or you can copy them from the CD supplied with AMTI Gen5.
2. Connect the AMTI Gen5 to the computer. The installer will ask for the drivers browse to the folder where you saved them. Alternatively, follow the instructions for how to install the AMTI drivers, which are included in the ZIP-file with the driver.
3. Connect a BNC cable from Sync out on the Oqus camera to the **Genlock/Trigger input** connection on the AMTI Gen5 amplifier. An adapter is needed from the BNC connector to the RCA connection of AMTI Gen5, this is supplied by Qualisys AB.

4. Connect the force plate to the AMTI Gen5 amplifier.



These are the hardware connections that are needed. The next step is to activate the AMTI Gen5 force plate in QTM.

1. Go to **Input Devices** page in the **Project options** dialog.
   a. Activate the AMTI Gen5 amplifier that are connected to the system.
2. Go to the page for each amplifier under the **Force plates** page, see chapter "AMTI Gen5" on page 178.
   a. Set the **Sync source** option to the device that the sync cable is connected to.
   b. Click on **Advanced** to change the frequency of the force plate. You will go to the **Synchronization** page with just the device used as Sync source selected. Change the **Synchronization output** setting to the desired frequency, see chapter "Synchronization output" on page 162. It is recommended to use the **Camera frequency multiplier** option to set the frequency, because for example when exporting to C3D then the force frequency must be a multiple of the marker capture rate.
      Note: If there is an analog board in the system, that is also frame synchronized, then it is recommended to use the same camera as synchronization source for the analog board and the AMTI Gen5. Then you can use the Sample rate option on the Analog board page to control the frequency on both of the boards.
3. Go to the **Force data** page in the **Project options** dialog. Rename the force plate if you want it to have a custom name.
4. Open the **Force plate** page for each AMTI Gen5 force plate.
   a. Enter the position of the force-plate, see chapter "Force plate location" on page 224. 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, check that the Sync cable is connected correctly.
   a. If the cable is connected, make a test in AMTI Netforce to see if there is force data in that program.

Note: The AMTI Gen5 plate is reset 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

Supported force plate devices

QTM supports calculation of force data from analog data from the force plates of the following manufacturers: AMTI, Bertec and Kistler. The amplifiers Kistler 5695A DAQ and the AMTI Gen5 and OPT-SC are integrated in QTM so that there is no need for an analog board.

For information about how to use the force data see chapter “Force data calculation” on page 386. 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 211.

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 “Appendix D: Wireless EMG systems”.

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 450.

Note: The channels must be activated on the page for the analog board in the **Project options** dialog, see chapter “Channels” on page 170. 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 video devices

Video capture

QTM can record video data, including the sound, together with the motion capture data. This data is only for documentation purposes and are not used in the calculation of 3D data. The video source can be either via DirectShow (Blackmagic Design cards or DV/webcam devices), see respectively chapters “Video capture with Blackmagic Design cards” below and “DV/webcam devices” on page 473.

The video capture via DirectShow 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 page 473.

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 “External video devices in 2D view” on page 36.

Video capture with Blackmagic Design cards

QTM supports the video capture cards Intensity Pro and DeckLink Mini Recorder from Blackmagic Design. It is PCI Express cards that can capture video directly from a video camera. The Intensity Pro card supports video from different types of video sources (e.g. HDMI, Component, S-video and Composite). The card can also capture the sound on the HDMI input or from an analog RCA input. The DeckLink Mini Recorder card supports video and sound from HDMI and SDI.

Installing the BlackMagic Design card

It is important to notice that you need one Blackmagic Design card for each video source that you want to capture. Therefore, you also need a PCI Express slot on the computer for each card. If you have a computer made before 2008 there might not be that many PCI Express slots on the motherboard. It is also required that the graphic board can handle hardware acceleration, which means that a motherboard integrated graphics card will not work. Contact Qualisys AB to make sure that it works on your computer.

Follow these steps to install the card in the computer:

1. Install the drivers (Desktop Video x.x.msi). It is recommended to download the latest version from www.blackmagic-design.com.
2. Shut down the computer.

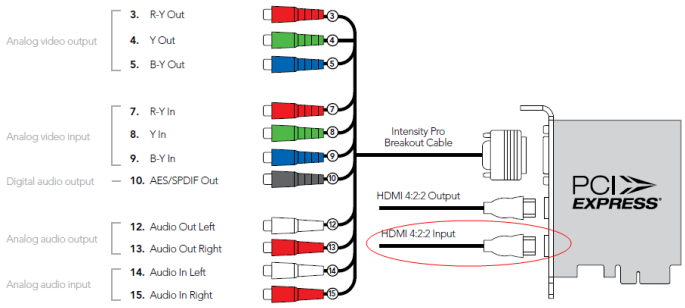
3. Install the card in a free PCI express slot.
4. Start the computer and click through the driver installation.
5. When the installation is finished, open the Desktop Video Setup application.
   a. Make sure that the input is set to **HDMI Video & HDMI Audio**, unless you are using another input.
   b. Close and click OK if there is an elevation question.

For more information about how to install the card see the manuals from Blackmagic Design.

Connecting a video source to the Intensity Pro card

The Intensity Pro card can capture from different types of video sources (HDMI, Component, S-video and Composite). The card can also capture the sound on the HDMI input or from an analog RCA input. However it can only capture from one of the video inputs at a time so you have to follow these instructions when connecting a video source.

1. Select the correct input for your video source on the **Blackmagic Control Panel** on the Windows control panel. You get the best image with **HDMI input**, but you can use that with either**HDMI audio** or **Analog RCA audio**. You do not have to change any other setting on the control panel.
2. Connect the video source to the input you have chosen in the **Blackmagic Control Panel**. You have to use the breakout cable for other inputs than HDMI.
   IMPORTANT: Make sure that you connect the HDMI cable to the HDMI input, see image below.
   IMPORTANT: If you are using the breakout cable, make sure that you read the labels on the connectors so that you use the input connectors and not the output connectors. For more information about the analog inputs see the manual from Blackmagic Design.



Connecting a video source to the Decklink Mini Recorder card

The Decklink Mini Recorder card can capture video and sound from either HDMI or SDI. However it can only capture from one of the video inputs at a time so you have to follow these instructions when connecting a video source.

1. Select the correct input for your video source on the **Blackmagic Control Panel** on the Windows control panel. On a standard camcorder the output is always HDMI, so set the video source to **HDMI input**. The SDI output is only available on pro camcorders.
   You do not have to change any other setting on the control panel.
2. Connect the video source to the input you have chosen in the **Blackmagic**

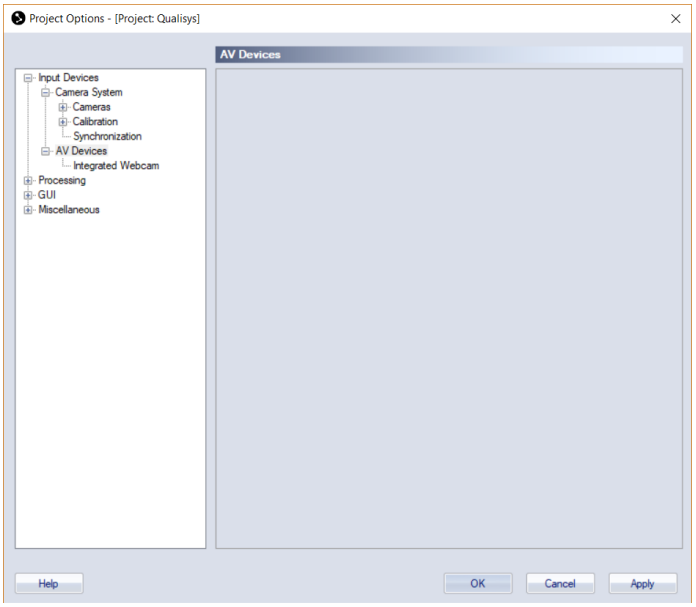
Control Panel.

Using Blackmagic Design video source in QTM

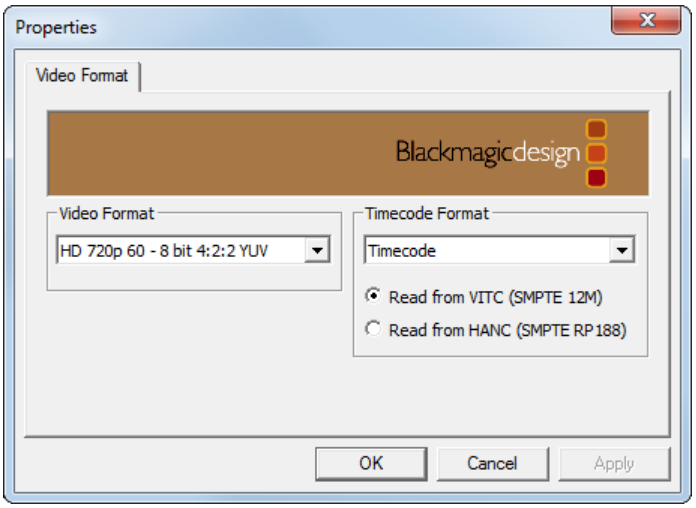
QTM uses DirectShow to capture the video and audio from the Intensity Pro card. It will, therefore, appear as a video device called Decklink Video Capture on the **Video Devices** page in **Project options**. To use the board follow the instructions below:

1. Before you start QTM you must select the correct input on the card on the **Blackmagic Control Panel** on the Windows control panel, see chapter "Connecting a video source to the Intensity Pro card" on the previous page or "Connecting a video source to the Decklink Mini Recorder card" on the previous page. HDMI usually gives you the best image. You do not have to specify the output or any of the other settings.
2. On the **Input Devices** page, select the checkbox in the **Enabled** column next to **Decklink Video Capture** option. If you have several cards it will be numbered starting from the top of PCI express slots.

   ⚠ **IMPORTANT:** Do not select the **BlackMagic WDM Capture** option it does not work in QTM.



3. Start a new measurement and open a 2D view window.
4. The Video view is most probably black, because the **Video format** is on the default value. Right-click on the Video view for the Blackmagic card and select **Video camera settings**.



5. Select the **Video Format** of the video source and click OK. You must check which format is actually used by the video camera, there is usually a separate setting for the HDMI output. If you select the wrong format the image will be black in QTM.

   Note: You have to use OK to see if the settings work, because **Apply** will not update the settings.

   The options for the **Video Format** goes from **NTSC** to **HD1080p 24**. The first five settings for PAL and NTSC are ambiguous, but they correspond to the following formats.

   NTSC

   NTSC 23.98

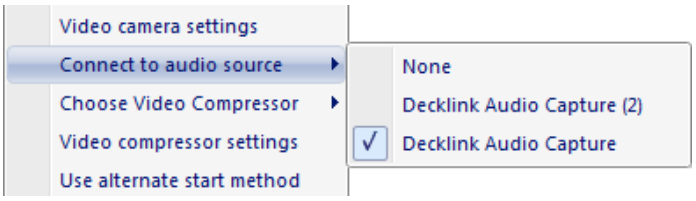
   PAL

   NTSC Progressive

   PAL Progressive

   The rest of the formats can be interpreted directly from the settings, e.g. **HD 720p 59.94**options is 720p at 59.94 Hz. If you use the HDMI input QTM will automatically detect the number of pixels in the image and scale the video view accordingly.

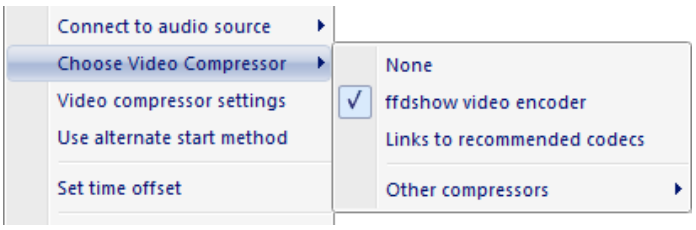
6. Then if you want to record sound with the video, right click on the Video view again and select the correct audio source on the **Connect to audio source** option. QTM will remember the audio source as long as the Intensity Pro card is installed on the computer.



7. It is recommended to use a codec to compress the video files, otherwise the files will be extremely large. The recommended codec is MJPEG in the FFDS package, because it is fast and has a reasonable compression. On the standard quality of 85 the video size is about 2-4 MB/s. It can also most of the time be played on a computer without the codec package installed.
   If you have a SSD disk on your computer then it is possible to capture

uncompressed video, but the file size can be around 40 MB/s.

To select the MJPEG codec right-click on the Video view and select **ffdshow video encoder** on **Choose video compressor**. You also have to make sure that you have selected the MJPEG codec in the **Video compressor settings**, see chapter "FFDS-MJPEG codec settings" on page 138. For more information about how to install the codec see chapter "Codecs for Oqus high-speed video files" on page 291.



8. Because the video is recorded with DirectShow the start of the video is not totally in synchronization with the marker data. It is therefore recommended that you look at the offset in a file and change it with the **Set time offset** option on the menu. The offset will usually be the same as long as you have the same measurement setup. For more information see chapter "Video offset" on page 473.

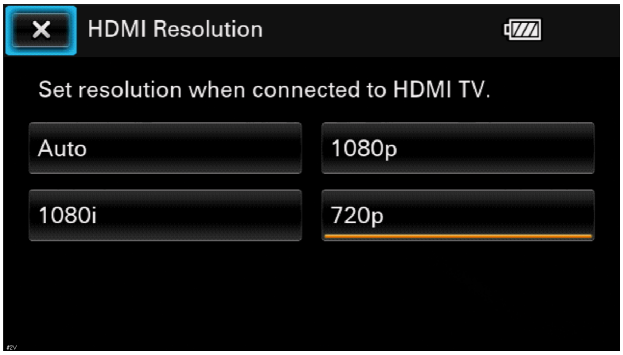
Settings on Sony HDR-CX330

On a new Sony HDR-CX330 camera you need to change the HDMI output and some other settings to make it work best with QTM and the Blackmagic Design card. Click on **Menu** with the control button to change the video settings and the settings menu below is displayed on the camera.

First of all you must set the HDMI resolution so that you know what Video settings to use in QTM.

- **HDMI Resolution**
  - Go to **Setup** and scroll down to **HDMI Resolution** and open that option.

  - Select the option that you want to use. The recommended option is **720p/480p** which will give you 720p 59.94 Hz.



The list below shows different resolutions and the matching **Video format** setting in QTM, for information about how to change the setting in QTM see chapter "Using Blackmagic Design video source in QTM" on page 466.

Note: If the video camera is a PAL model the frequency changes from 59.94 Hz to 50 Hz and NTSC to PAL.

Auto - Unknown

1080p -
Not supported by Intensity Pro because the frequency is too high.

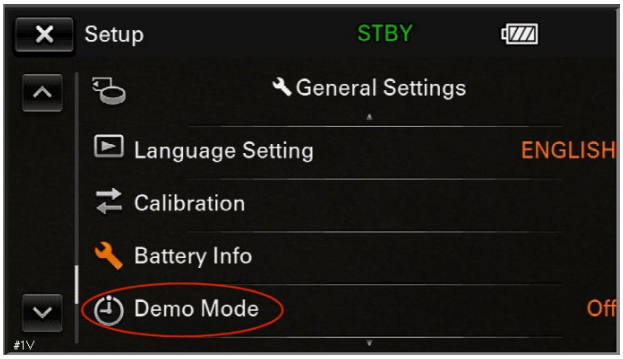
1080i - HD 1080i 59.94 - 8 bit 4:2:2 YUV
This option is interlaced which is not recommended when being viewed on a computer, because when played there will be horizontal lines in the image.

720p - HD 720p 59.94 - 8 bit 4:2:2 YUV
Recommended option because it is the highest possible resolution that uses progressive scanning. The image is good, but the files will be large so it is recommended to compress the files in QTM.

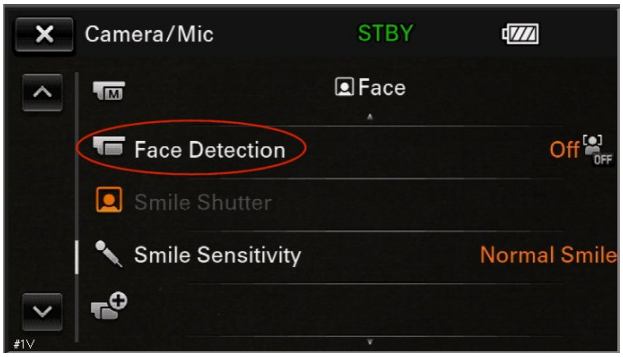
Then it is recommended to change the two settings that makes the camera operate better.

- **Demo mode**
  Turn off **Demo mode** on the camera, otherwise the camera will start showing you demo pictures after a while.

- Go to **Setup** and scroll down in the list until you find the **Demo mode** option and turn it off.



- **Face detection**
  - Go to **Camera/Mic** on the first menu and scroll down to the **Face Detection** option and turn it off. If not turned off there will be a rectangle in the image as soon as a face is detected.



Settings on Sony HDR-CX430V

On a new Sony HDR-CX430V camera you need to change the HDMI output and some other settings to make it work best with QTM and the Blackmagic Design card. Click on **Menu** on the top left corner on the touch screen to change the video settings and the settings menu below is displayed on the camera.

First of all you must set the HDMI resolution so that you know what Video settings to use in QTM.

- **HDMI Resolution**
  - Go to **Setup** and scroll down to **HDMI Resolution** and open that option.

  - Select the option that you want to use. The recommended option is **720p/480p** which will give you 720p 59.94 Hz.

The list below shows different resolutions and the matching **Video format** setting in QTM, for information about how to change the setting in QTM see chapter "Using Blackmagic Design video source in QTM" on page 466.

Note: If the video camera is a PAL model the frequency changes from 59.94 Hz to 50 Hz and NTSC to PAL.

Auto - Unknown

1080p/480p -
Not supported by Intensity Pro because the frequency is too high.

1080i/480i - HD 1080i 59.94 - 8 bit 4:2:2 YUV
This option is interlaced which is not recommended when being viewed on a computer, because when played there will be horizontal lines in the image.

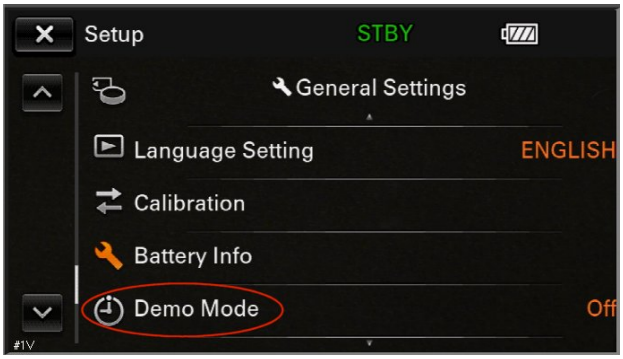
720p/480p - HD 720p 59.94 - 8 bit 4:2:2 YUV
Recommended option because it is the highest possible resolution that uses progressive scanning. The image is good, but the files will be large so it is recommended to compress the files in QTM.

480p - NTSC - 8 bit 4:2:2 YUV(The fourth item in the list in QTM)
This is the same resolution as the American analog TV resolution, but it uses progressive scan so there are no lines in the image. You can use this option if you want smaller video files than **720p/576p**.

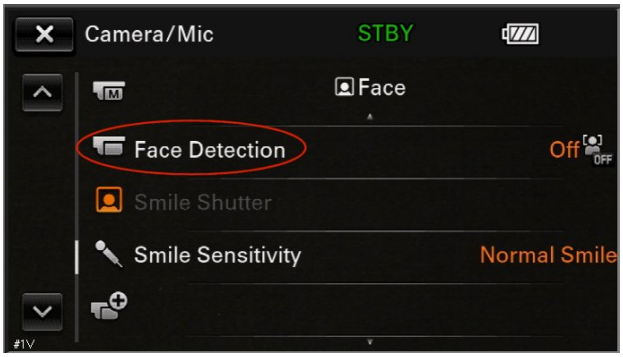
480i - NTSC - 8 bit 4:2:2 YUV(The top option in the list in QTM)
This is the American TV analog resolution. It is interlaced and has a low resolution.

Then it is recommended to change the two settings that makes the camera operate better.

- **Demo mode**
  Turn off **Demo mode** on the camera, otherwise the camera will start showing you demo pictures after a while.
  - Go to **Setup** and scroll down in the list until you find the **Demo mode** option and turn it off.



- **Face detection**
  - Go to **Camera/Mic** on the first menu and scroll down to the **Face Detection** option and turn it off. If not turned off there will be a rectangle in the image as soon as a face is detected.



Panasonic AW-HE2

To setup the Panasonic camera for use with QTM, connect the power adapter and an HDMI cable. Refer to the Panasonic manual for how to connect the power adapter and switch on the camera. The LED on the front of the camera should light green when the camera is switched on.

Connect the Panasonic AW-HW2 camera with a standard HDMI cable to the input of the BlackMagic design card. Follow the instruction under “Using Blackmagic Design video source in QTM” on page 466 to enable video capture in QTM.

For Video format (step 5 in “Using Blackmagic Design video source in QTM” on page 466) use **HD 720p 59.94 Hz - 8 bit 4:2:2 YUV**.

DV/webcam devices

Only DV/webcam devices that support the Microsoft DirectShow API can be used with the QTM software. This is true for most webcams, however there are very few video cameras equipped with a DV–output or FireWire 4-pin connection. This is a requirement to be able to stream out video in real-time. The option for video cameras without firewire is to use the Intensity Pro card, see chapter “Video capture with Blackmagic Design cards” on page 464.

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 the next page.

The video devices are connected to QTM on the **Input devices** page in the **Project options** dialog, see chapter “Input Devices” on page 121. For the DV cameras the audio is recorded directly with the video. Webcams on the other hand has the audio sent separately to the computer, which means that you have to select it manually, see chapter “Selecting audio source” on the next page.

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 291. 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 138.

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 on external software.

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.

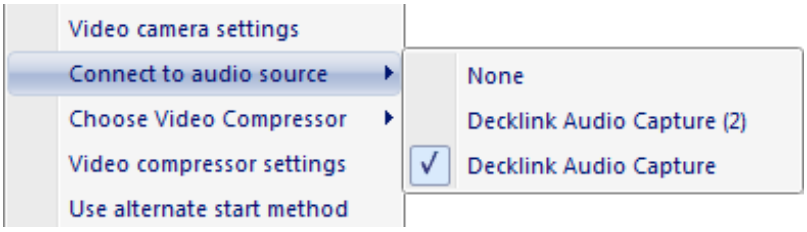
2. Then click on **Set time offset** and enter the starting time of the video in the dialog. The starting time is in the measurement time, i.e. with a 1 s starting time the video file will start at 1 s when looking in the 3D view window.
3. Play the file and check that it is ok.
   Note: Because of the 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.

Selecting audio source

QTM can record the audio from DirectShow devices together with a DirectShow video source. This can for example be the microphone on the computer or audio on the Intensity Pro card. The available audio sources is listed in the menu, however for DV cameras this option is not available because the audio is already included in the video.

To set the audio source for a video source follow these instructions.

1. Start a new measurement and open a 2D view window.
2. Right-click on the Video view that you want to set the audio source for and click on **Connect to audio source** in the menu.



3. Then select the audio source you want from the list. QTM will remember the audio source as long as the video source is available on 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” on the previous page.

Timing hardware

How to use external trigger

An external trigger can be used to trigger the start of the motion capture, see chapter "Trigger ports" on page 156. 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.

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.

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.

When using analog boards with pretrigger the connection differs between the boards, see below.

The pretrigger settings can be found on the **Synchronization** page in the **Project options** dialog, see chapter "Pretrigger (Oqus)" on page 158.

Measurement with analog capture while using pretrigger (Oqus)

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 **Synchronization** 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
Use normal setup with a **Sync out** signal from one of the Oqus cameras connected to the **Sync input** of the analog device, see chapters "Connection of analog board" on page 450. In addition, the external trigger signal should be connected to the first analog channel on the board. This can be done by splitting the trigger input at the camera or sync unit with a T-connector and connect one end to the trigger

button and the other end via a BNC cable to the first analog channel on the board.

Note: 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.

Warning: The synchronization of the analog data may not exactly coincide with the start of the camera frames, as the time of the trigger start signal may fall anywhere in a pretrigger camera frame. This results in a latency within one camera frame period, which varies between captures.

How to use external timebase

The external timebase is used to get a required frame rate in the motion capture from an external device.

The Qualisys camera system can be synchronized with an external timebase for drift free synchronization with a external devices. When synchronizing to a periodic TTL signal the Qualisys system will lock the capture frequency to that of the external signal source, see chapter “Using External timebase for synchronization to a periodic TTL signal” below for detailed information.

It is also possible to use a time code signal as an external timebase source. Currently, two standards are supported, SMPTE (requires an Oqus or Camera Sync Unit) and IRIG (requires a Camera Sync Unit). For more information, see chapter “Using External timebase for synchronization to a time code signal” on page 478.

An external timebase can also be used to trigger individual camera frames, using non-periodic signal mode. For an example of how to synchronize the camera system with burst signals, see chapter “External timebase with bursts of signals with constant period (cycle) time and with delays between the bursts” on page 479.

The settings for external timebase are on **Synchronization** page in the **Project options** dialog, see chapter “External timebase” on page 159.

External timebase connection

For an Oqus system the external timebase is connected to the sync in connector. Contact Qualisys AB to get a special kind of splitter cable with this connector.

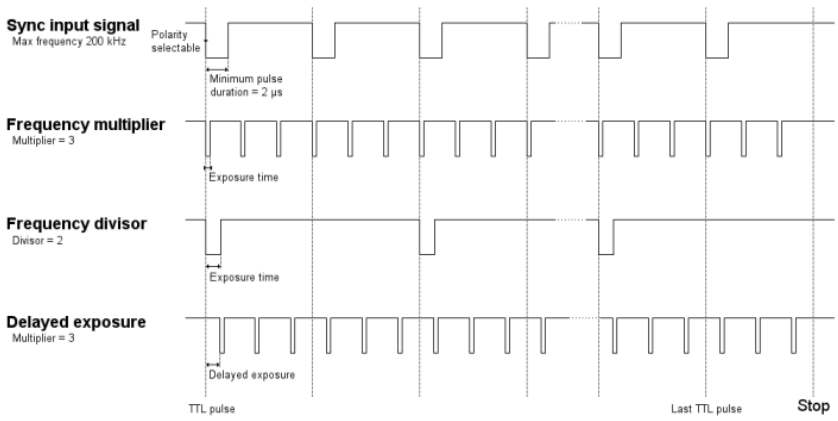
Note: If an external trigger is used to start the measurement, it must be connected to the same camera as the external timebase.

For a Miqus system the external timebase is connected to a Camera Sync Unit.

Using External timebase for synchronization to a periodic TTL signal

The synchronization input of Qualisys camera systems can be used to synchronize the frames of the camera capture to an external source. This means that the camera frequency will depend on the input signal so that there is no drift between the two systems.

When synchronizing a Qualisys system to a period signal, set the **Signal Mode** under the **External timebase** options to **Periodic**. The synchronization signal from the external hardware must be a periodic TTL signal between 0 and 5 Volt. You can use a frequency multiplier or divisor to get the camera frequency that you need. For detailed information about the External timebase settings, see chapter “External timebase” on page 159.



For Oqus systems, the synchronization signal can be connected to the **Sync in** port of any of the cameras within the system. For Miqus systems, the synchronization signal is connected to the **Sync** input port of a **Camera Sync Unit**. It is important that the synchronization signal is continuously sent to the camera so that the synchronization is not lost.

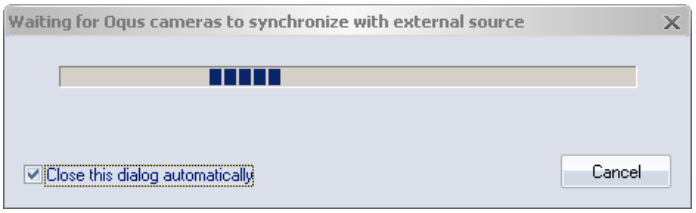
Follow these instructions to use external timebase with a periodic signal.

Real-time

1. Make sure that the settings for the **External timebase** on the **Synchronization** page in the **Project options** dialog are correct for your setup.

   Warning: Do not use **External timebase** settings that result in a camera frequency higher than 0.975*maximum frequency at the full image size.

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

1. 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 **Synchronization** page.

Using External timebase for synchronization to a time code signal

The Qualisys system can be synchronized to an external time code. The following standards are supported:

- SMPTE at frame rates of 24 Hz, 25 Hz, and 30 Hz, without dropped frames.
- IRIG A and B, Direct Current Level Shift (DCLS)

SMPTE

SMPTE time code is a set of cooperating standards to label individual frames of video or film with a time code defined by the Society of Motion Picture and Television Engineers. SMPTE time code signals can be generated by time code generators or external audio interfaces. SMPTE is commonly used for synchronization in audio/video applications. Qualisys supports SMPTE standards at frame rates of 24 Hz, 25 Hz and 30 Hz, without dropped frames. For more information about SMPTE time code, see https://en.wikipedia.org/wiki/SMPTE_timecode. For more specific information about how to use SMPTE for synchronization of captures with audio recordings, see chapter "Using SMPTE for synchronization with audio recordings" on page 488.

The use of SMPTE time code requires an Oqus or Camera Sync Unit. For both devices, the SMPTE time code signal should be connected to the **SMPTE** input port. The capture frequency can be set to a multiple or a divisor of the SMPTE frame rate in the **External timebase** settings on the **Synchronization** page.

Warning: Do not use **External timebase** settings that result in a camera frequency higher than 0.975*maximum frequency at the full image size.

Note: When using SMPTE as an external timebase source the shutter delay (**Delay from signal to shutter opening** on the **External timebase** page) should be at least 100 microseconds for a consistent number of cameras frame per SMPTE frame. The shutter delay value is automatically set to 100 µs when selecting SMPTE.

IRIG

Inter-Range Instrumentation Group time codes, commonly known as IRIG time codes, are standard formats for transferring timing information. Atomic frequency standards and GPS receivers designed for precision timing are often equipped with an IRIG output. IRIG is commonly used in communication systems, data handling systems, military systems, telemetry systems, high speed cameras and in machine vision. Qualisys supports the IRIG A and the IRIG B standards of DCLS type (Direct Current Level Shift). For more information about IRIG time code, see https://en.wikipedia.org/wiki/IRIG_timecode.

The use of IRIG time code requires a Camera Sync Unit. The IRIG time code signal should be connect to the **Event** input port on the Camera Sync Unit. The capture frequency can be set to a multiple or a divisor of time code frequency in the **External timebase** settings on the **Synchronization** page.

Warning: Do not use **External timebase** settings that result in a camera frequency higher than 0.975*maximum frequency at the full image size.

Timestamps

When using time code as external timebase source, time stamps are recorded by default, so that each camera frame includes a time stamp. When the capture rate is set to a multiple or divisor of the time code frequency, the first camera frame of a capture may not correspond to the start of a time code period or frame. For synchronization to an external signal you will need to look up a camera frame at which the time code increased and take the offset into account.

Using Qualisys video with External timebase

Both Oqus and Miqus video can be used with External timebase, but only if the video capture rate is a divisor of the marker capture rate. The video capture rate also needs to be an integer so you cannot use any divisor on the marker capture rate. The presets that can be used are activated automatically for the current marker capture rate. For example, if the marker capture rate is 200 Hz, you can capture video in 200, 100, 50, 40, 25 and 20 Hz. These are the divisors 1, 2, 4, 5, 8, and 10.

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**Synchronization** 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** headingin 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 0.975*maximum frame rates at full FOV are the following.

Oqus	Max frequency Standard mode	Max frequency High speed mode
1-series	240 Hz	-
2c-series	328 Hz	649 Hz
3-series	489 Hz	-
3+-series	489 Hz	1719 Hz
4-series	472 Hz	-
5-series	175 Hz	-
5+-series	175 Hz	346 Hz
7+-series	292 Hz	1092 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.

The camera will be damaged if it is overclocked and the guarantee is then void.

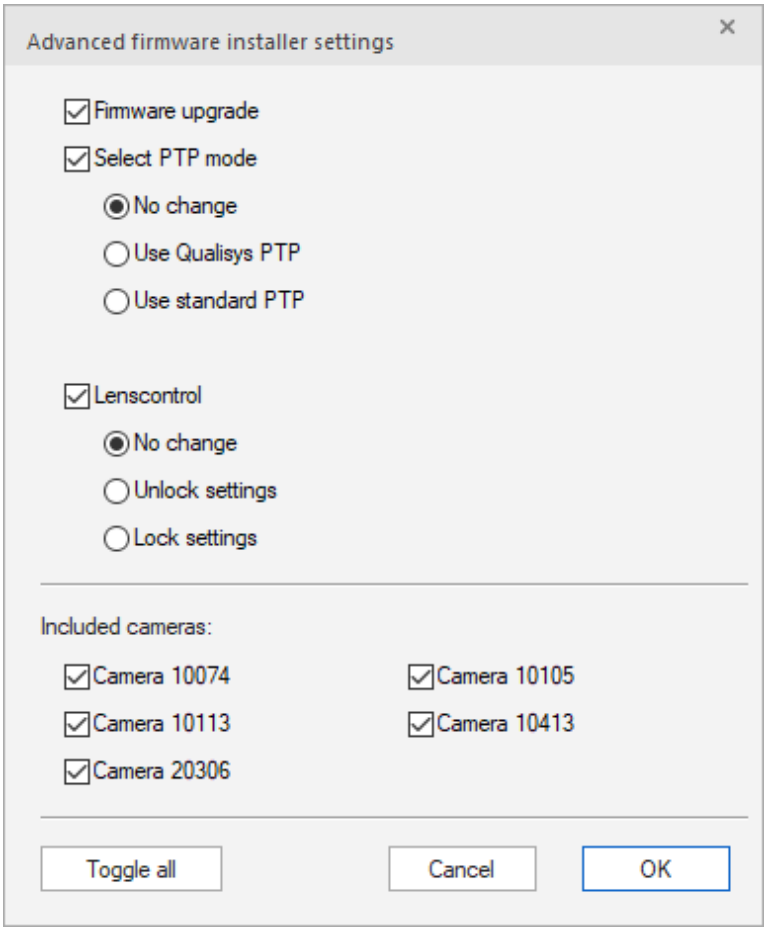
Note: The tracking can become significantly of poorer quality if you use a timebase with a large difference between the smallest and the largest delay between two frames. Since the tracking assumes a constant time period between two frames.

How to use PTP sync with an external clock master (Camera Sync Unit)

The Camera Sync Unit (CSU) can be used with any Qualisys camera system and configured to synchronize with an external clock master using the standard PTP protocol. Follow the steps outlined below to set up the camera system.

1. Start the correct version of the Qualisys Firmware Installer (QFI.exe), see “Appendix B: Qualisys Firmware Installer (QFI)” for more information.
2. Click on the **Advanced** button.

3. Under Select PTP mode, select **Use standard PTP**. The **Firmware upgrade** option can be deselected to speed up the process.



4. Click OK and continue installing the firmware.

Connecting to an external clock master

The external clock master should be connected to the same Local Area Network as the cameras. The following requirements apply to the external clock master:

- Standard: IEEE 1588:2008 (PTPv2)
- ipv4/udp
- Delay mechanism: End-to-End (E2E)
- Two-step clock
- Sync message interval: 1s

For adding the timestamp to the motion capture frames, activate the timestamp option on the **Synchronization** page under **Project Options** and set the type to **Camera time**, see chapter “Timestamp” on page 162.

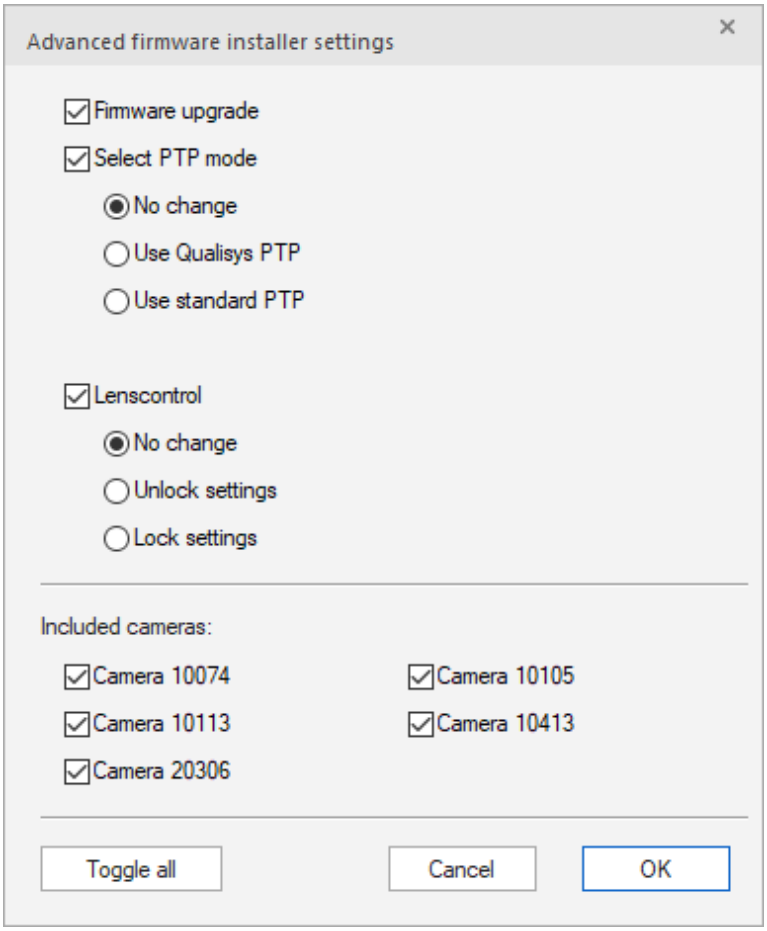
Note: Arqus and Miqus systems can use PTP sync without a Camera Sync unit, but then the timestamps can't be added to the capture frames.

Note: Oqus systems can use PTP sync without a Camera Sync unit, but it requires extra configuration, see chapter “How to use PTP sync with an external clock master (Oqus)” on the next page.

How to use PTP sync with an external clock master (Oqus)

Oqus systems can be configured to synchronize with an external clock master using the standard PTP protocol. Follow the steps outlined below to set up the camera system.

1. Start the correct version of the Qualisys Firmware Installer (QFI.exe), see “Appendix B: Qualisys Firmware Installer (QFI)” for more information.
2. Click on the **Advanced** button.
3. Under Select PTP mode, select **Use standard PTP**. The **Firmware upgrade** option can be deselected to speed up the process.

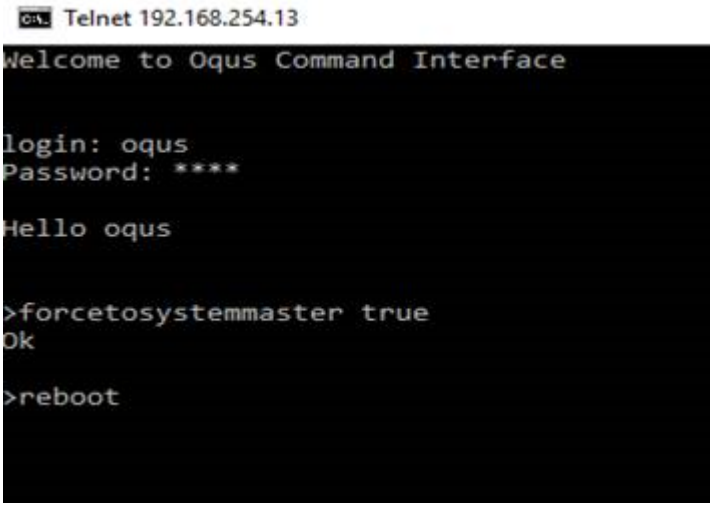


4. Click OK and continue installing the firmware.

For Oqus systems without a **Camera Sync Unit**, one of the cameras need to be manually set as system master. To manually set the system master, follow these steps.

1. Start a telnet client and log in to the selected camera (login: oqus; password: oqus)
2. Type the command **forcetosystemmaster true** and press enter.
3. Type the command **reboot** and press enter.
4. Wait for the camera to reboot.

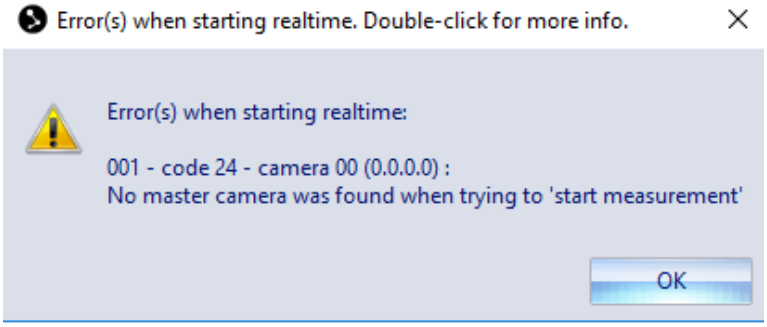
5. Type the command **exit** and press enter to quit the telnet session.



Note: It is recommended to use the camera that is the master camera when there is no external clock master connected to the system, so that the system will work even if the external clock master is removed from the system.

The following should be kept in mind for the system master:

- A camera that is PTP master is also system master.
- A camera that has been set to system master with the forcetosystemmaster command will always be system master.
- There must be exactly one system master.
- If no system master is detected, it is not possible to start a measurement. The following error message will be displayed.



Connecting to an external clock master

The external clock master should be connected to the same Local Area Network as the cameras. The following requirements apply to the external clock master:

- Standard: IEEE 1588:2008 (PTPv2)
- ipv4/udp
- Delay mechanism: End-to-End (E2E)
- Two-step clock
- Sync message interval: 1s

For adding the timestamp to the motion capture frames, activate the timestamp option on the **Synchronization** page under **Project Options** and set the type to **Camera time**, see chapter “Timestamp” on page 162.

Note: For Oqus systems, the camera timestamp in exported files needs to be converted as the number of bits in which the timestamp is stored is limited to 48. It is recommended to add a **Camera Sync Unit** to the system to avoid the need to convert the timestamps. Contact Qualisys support of you need more information about this conversion.

Synchronizing external hardware

How to synchronize external hardware

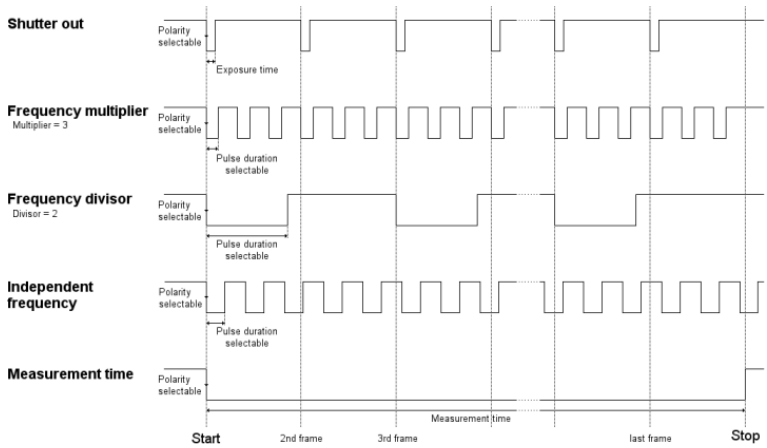
It is possible to synchronize other external hardware that are not directly integrated in QTM. This can be achieved either by triggering the external hardware from the cameras or sending a signal from the external hardware to the cameras. Which option to use depends on the specification of the external 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 162, 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.



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.

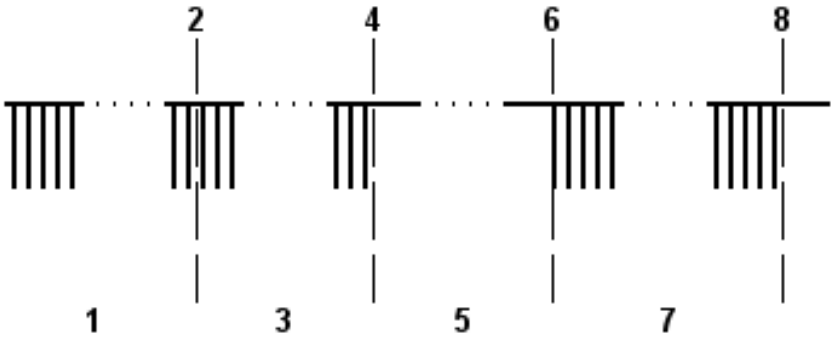
The external capture can also be stopped by the camera system if the **Frequency multiplier** mode is used and the pulse duration is the same as the period time.

Then the signal will for example be low during the whole measurement until it stops, see chapter “Synchronization output” on page 162.

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 162.

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.

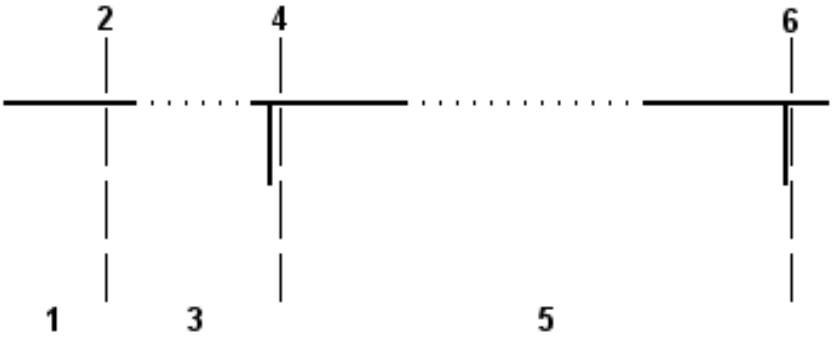


1. **Preview**
   During preview a pulse is sent on each preview frame.
2. **Capture**
   Click on **Capture** in the **Capture** menu.
3. **Start capture dialog open**
   When the **Start capture** dialog is open the pulse continues because the preview is still being updated.
4. **Start**
   Click on **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.
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.
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 Oqus sync unit for synchronization

By connecting the Oqus sync unit to the master camera in the Oqus system, you can use an SMPTE signal to timestamp and synchronize the measurements. The sync unit can also handle a video signal (black burst or tri-level) for synchronization (also called genlock). For more information about how the synchronization works in sound

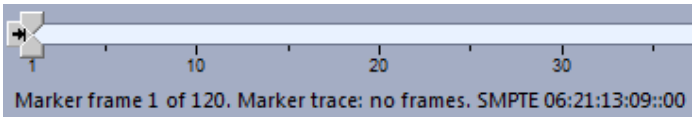
applications see chapter “Using SMPTE for synchronization with audio recordings” on the next page.

The sync unit has the following opto-isolated connections.

Trig in/SMPTE in
This connection is used either as the standard Trig in connection or to convert the SMPTE signal to a signal compatible with the Oqus camera. The SMPTE signal can be used for both timestamp and synchronization. The LEDs next to connector will indicate which input you are using in QTM.

Note: For the best detection of the SMPTE timecode use one of the Analog outputs on the Motu device so that you can increase the amplitude of the signal. Move the dial in the SMPTE program so that it is pointing right to get a high enough amplitude. The SMPTE timecode output on the MOTU device is often too low for the Sync unit to get a stable detection of the SMPTE timecode.

Timestamp
To activate the SMPTE timestamp open the **Synchronization** page in the Project options dialog. Then select **Use SMPTE timestamp** below the **SMPTE** heading also make sure that you select the correct **SMPTE frequency**, see chapter “Timestamp” on page 162.



The timestamp is displayed in the **Timeline control** bar. Because the timestamp is sampled at for example 30 Hz there can be several frames per timestamp. Which is indicated by the last number (subframes) after '::' in the timestamp.

Synchronization
To activate the SMPTE synchronization open the **Synchronization** page in the **Project options** dialog. Then select **SMPTE** as **Signal source** below the **SMPTE** heading, see chapter “External timebase” on page 159.
Note: Remember to select the correct **SMPTE frequency** under the **SMPTE** heading.

Sync in/Video in
The connector can be used either for Sync in or Video in synchronization. The LEDs next to connector will indicate which input you are using in QTM.

Sync in
The Sync in connection is standard connector for an external timebase

connection, see chapter “External timebase” on page 159 and “Using External timebase for synchronization to a periodic TTL signal” on page 476. For example the word clock of up to 48 kHz from a sound sampling board can be connected to this connection and then divided to the wanted frequency on the **Synchronization** page in the **Project options** dialog. Make sure that you have set a divisor that gives a camera frequency that works with the Oqus camera.

Note: If you use SMPTE for timestamp and another external signal for synchronization, both must be connected to the same camera.

Video in
To activate the video synchronization open the **Synchronization** page and select **Video sync** on the **Control port** setting, see chapter “External timebase” on page 159. When the **Video in** option is selected the sync unit will decode a black burst or tri-level video signal so that the Oqus camera can lock in on the signal. Just as for the standard sync-in you need to have time reference to know when the measurement starts, for example you can use the SMPTE signal.

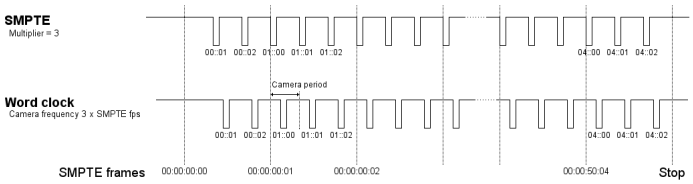
Sync out
This is the standard Sync out connection from the camera, see chapter “Synchronization output” on page 162 and “Using Sync out for synchronization” on page 484.

Using SMPTE for synchronization with audio recordings

The Qualisys system can be synchronized with audio recordings by using the SMPTE timestamp via the Oqus or Camera Sync Unit. QTM has been tested with the MOTU 828mk3, but in theory any equipment with an SMPTE signal can be used. Check the following information to synchronize QTM with audio recordings. For information about the settings for external timebase see chapters “External timebase” on page 159 and “Timestamp” on page 162.

1. It is important to enter the nominal frequency and the SMPTE frequency on the **Synchronization** page in the **Project options** dialog. Otherwise, the data cannot be compared correctly with the audio recordings.
2. For Oqus systems, make sure to connect the Oqus sync unit to the master camera, the camera with an M in the display, see chapter “Using Oqus sync unit for synchronization” on page 486.
3. If you want a sync output frequency from the camera system above 5000 Hz, you must use the master camera.
4. **Synchronizing with Word clock or SMPTE**
   In theory the Word clock should be better to use as external timebase than SMPTE. However, the SMPTE signal from the MOTU unit is stable enough for the camera synchronization and it has the advantage that the marker capture will start without a variable offset to the SMPTE time code signal.
   a. If you use SMPTE as the sync source then the start of the measurements will be aligned with the SMPTE frames, see image below.
      Note: If you use SMPTE as external timebase it is important that you do not use a too large divisor so that the resulting frequency becomes lower than 2 Hz. This will result in the camera not being able to synchronize to the signal.

b. When using Word clock it is important that you also use the SMPTE clock for reference of the start of the measurement. However, alignment of the camera frames relative to the SMPTE frames will not be exact and there will be a variable delay of up to one camera frame. For example at 200 Hz the maximum start offset is 5 ms, see image below. This is because the camera can start at any time within an SMPTE frame and then the camera frame will not be aligned with the SMPTE signal.



5. It is recommended to use a multiple of the SMPTE signal as camera frequency. This makes the data easier to compare to other systems. The data is synchronized even if it is not a multiple, but then you will have to take into consideration that the number of camera frames per SMPTE frame will vary.
6. The camera frequency with external timebase cannot be higher than the maximum for the camera at full FOV, for example 500 Hz for the 3-series camera or 1764 Hz with high speed sensor mode in Oqus 3+-series. This is because the preview mode uses the full image size of the camera.

To be able to compare the QTM data with the audio data it is important that the audio sequencer in some way records the SMPTE time code signal. This can be done in the following 3 ways with the MOTU 828mk3.

1. When using the MOTU device on a Mac you can use a special setting called Generate from sequencer in the MOTU SMPTE console. With this option the SMPTE signal is generated based on the time position in the MOTU AudioDesk sequencer software used for recording audio. However, this requires that you start the recording in AudioDesk before you can start preview in QTM and that the recording must keep running while the cameras are capturing data. Then it is important to select the BWF format for the files in AudioDesk so that the start of the SMPTE time can be read from the WAV file header. For detailed information about the BWF standard, see https://tech.ebu.ch/docs/tech/tech3285.pdf.
2. On Windows you must connect another SMPTE source to the MOTU unit and use a program that supports ASIO synchronization (e.g. Cubase), if you want to see the same SMPTE time in the program as in QTM and save audio files in BWF format with the correct timestamp.
3. The third alternative works on all sequencers and that is to record the SMPTE signal on a parallel audio track. However, in this case you must decode the SMPTE signal to synchronize the audio files. For Matlab we have a solution available for pairing and cropping audio files to mocap files using a SMPTE decoder made in cooperation with Javier Jaimovich at MuSE. Contact Qualisys AB if you are interested.

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.
- 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 network interface has the correct settings, see “Network card setup” on page 437.
- 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

The 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 437. Run the QDS wizard if you do not know how to change the IP-address settings.

- For wireless Oqus systems, check that the computer supports the set up of a hosted network, and that it 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 configure it for wireless use, see chapter “Setup Oqus system for wireless communication” on page 435.
- 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 254.

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 279.

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.
- 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 258.

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 258.

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 434.

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 **Synchronization** 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 327.

  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 279.

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 184.
- Make a new calibration.
- Check that the camera positioning is OK, see chapter “Camera positioning” on page 254.
- 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 184.

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 364.

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 329.

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 297 and “How to use auto marker masking” on page 298.

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 211.

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** 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 supports the set up of a hosted network. This requires a computer running Windows 7. This feature is no longer supported for most wireless adapters in Windows 10.
- Make sure that you use a special version of QDS that supports Oqus wireless camera setup. Contact support@qualisys.com if QDS does not show the Oqus wireless camera setup wizard when starting it.
- 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 437.
- 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 to setup a wireless connection for Oqus. This requires a special version of QDS. Contact support@qualisys.com if QDS does not show the Oqus wireless camera setup wizard when starting it.
- 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 is slow and some graphics are not showing.

Resolution

- Try updating the graphics drivers.
- Laptop with high-performance graphics (ATI or NVIDIA) and a Intel integrated graphics board.
  The computer does not automatically use the high performance board for QTM. Change the settings for the graphics board so that it always uses the high-performance graphics. For NVIDIA it works to just specify that QTM will use it. For ATI it seems like you have to switch the whole computer to ATI to be sure that it is using QTM.

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: Qualisys camera manual

This appendix describes the Qualisys cameras from a technical perspective. It describes the mechanical properties as well as communication interfaces and accessories. If you need more information, which is not covered in this reference, contact Qualisys support at support@qualisys.com.

Overview of camera models and sensor specifications

Qualisys camera sensor specifications (marker mode)

The below table gives an overview of the sensor specifications and sensor modes for all Qualisys camera models, when in marker mode.

Camera model	Sensor mode	Max capture rate (Hz)	Resolution
Arqus A5	5 MP @ 700 Hz	700	2560 x 1920
	1 MP @ 1400 Hz	1400	1280 x 960
Arqus A12	12 MP @ 300 Hz	300	4096 x 3072
	3 MP @ 1040 Hz	1040	2048 x 1536
Miqus M1	N/A	250	1216 x 800
Miqus M3	2 MP @ 340 Hz	340	1824 x 1088
	0.5 MP @ 667 Hz	667	912 x 544
Miqus M5	4 MP @ 183 Hz	183	2048 x 2048
	1 MP @ 359 Hz	359	1024 x 1024
Miqus Hybrid	2 MP @ 340 Hz	340	1824 x 1088
	0.5 MP @ 667 Hz	667	912 x 544
Miqus Video (Mono/Color)[1]	N/A	340	1920 x 1088
Oqus 1	N/A	247	640 x 480
Oqus 2c[1]	2 MP @337 Hz	337	1920 x 1088
	0.5 MP @ 666 Hz	666	960 x 544
Oqus 3	N/A	503	1280 x 1024
Oqus 3+	1.3 MP @ 502 Hz	502	1296 x 1024
	0.3 MP @ 1740 Hz	1740	648 x 512
Oqus 4	N/A	476	1696 x 1710
Oqus 5	N/A	180	2352 x 1728
Oqus 5+	4 MP @ 179 Hz	179	2048 x 2048
	1 MP @ 355 Hz	355	1024 x 1024
Oqus 6+	6 MP @ 460 Hz	460	3072 x 1984
	1.5 MP @ 1662 Hz	1662	1536 x 992
Oqus 7+	12 MP @ 300 Hz	300	4096 x 3072
	3 MP @ 1121 Hz	1121	2048 x 1536

[1]Only in Marker intensity mode.

Oqus high-speed video sensor and memory specifications

The below table gives an overview of the sensor specifications, sensor modes and video buffer capacity for all Oqus high-speed cameras, when in video mode. The specifications are only applicable to uncompressed video. Note that the number of frames that can be stored in the buffer can be increased by cropping the image, see chapter “High-speed video” on page A - 6.

Camera type	Sensor mode	Max capture rate (Hz)	Resolution	Max video buffer capacity (frames)	Max video buffer capacity (seconds @ max capt. rate)
Oqus 1	N/A	247	640 x 480	3800	15.2
Oqus 2c	2 MP @ 337 Hz	337	1920 x 1088	540	1.6
	0.5 MP @ 666 Hz	666	960 x 544	1080	
	0.13 MP @ 1300 Hz	1300	480 x 272	2100	
Oqus 3	N/A	503	1296 x 1024	900	1.8
Oqus 3+	1.3 MP @ 502 Hz	502	1296 x 1024	900	1.8
	0.3 MP @1740 Hz	1740	648 x 512	3600	2.0
Oqus 5	N/A	180	2352 x 1728	290	1.4
Oqus 5+	4 MP @ 179 Hz	179	1920 x 2048	280	1.6
	1 MP @ 355 Hz	355	960 x 1024	560	
	0.25 MP @ 694 Hz	694	480 x 512	1100	
	0.06 MP @ 1328 Hz	1328	240 x 256	2200	

Note: The Oqus 5+ series image size is reduced for uncompressed video because of limitations on the data speed.

Qualisys video sensor specifications (in-camera MJPEG)

The below table gives an overview of the sensor specifications and sensor modes for all Qualisys video camera models when using in-camera MJPEG compression.

Camera type	Sensor mode	Max capture rate (Hz)	Resolution
Oqus 2c	2 MP @ 24 Hz	24	1920 x 1088
	0.5 MP @ 62 Hz	62	960 x 544
	0.13 MP @204 Hz	204	480 x 272
Oqus 5+	4 MP @ 13 Hz	13	2048 x 2048
	1 MP @ 50 Hz	50	1024 x 1024
	0.25 MP @ 176 Hz	176	512 x 512
	0.06 MP @ 557 Hz	557	256 x 256
Oqus 6+	6 MP @ 6 Hz	6	3072 x 1984
	1.5 MP @ 20 Hz	20	1536 x 992
Oqus 7+	12 MP @ 3 Hz	3	4096 x 3072
	3 MP @ 10 Hz	10	2048 x 1536
Miqus M1	N/A	30	1216 x 800
Miqus M3	N/A	30	1824 x 1088
Miqus M5	N/A	30	2048 x 2048
Miqus Hybrid	N/A[1]	86	1920 x 1088
Miqus Video (Mono/Color)	N/A[1]	86	1920 x 1088

[1]When using a resolution of 540p or lower, the Miqus Video camera automatically switches to a high-speed sensor mode.

Note: For information about maximum capture rate at different presets for the Miqus video/hybrid and Oqus 2c see chapter “Maximum capture rate for streaming video” on page 288.

Oqus technical reference

This section describes the Oqus camera from a technical perspective. It describes the mechanical properties as well as communication interfaces and accessories. If you need more information, which is not covered in this reference, contact Qualisys support at support@qualisys.com.

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 seven different image sensors, with different key features. Some provides exceptional resolution of 12.0 Mega pixels @ 150fps and

another exceptional speed of 500 fps at 1.3 Mpixels. The Oqus 2c-series camera is only available as an Oqus video camera with color sensor.

The position resolution of markers is defined in subpixels, which is obtained by multiplying the number of pixels in each dimension by a factor 64.

The table below shows how much the FOV must be reduced at different frame rates.

Frame rate [fps]	Oqus 1-series FOV	Oqus 2c-series FOV	Oqus 3-series and 3+-series FOV	Oqus 4-series FOV
50	Full resolution (640x480, 307200 pixels)	Full resolution (1920x1088, 2088960 pixels)	Full resolution (1280x1024, 1310720 pixels)	Full resolution (1696x1710, 2900160 pixels)
100	Full	Full	Full	Full
200	Full	Full	Full	Full
250	Full (247 Hz)	Full (337 Hz)	Full	Full
500	50%	67%	Full (502 Hz)	Full (485 Hz)
1000	25%	34%	50%	49 %
2000	N/A	17%	25%	24 %
5000	N/A	7%	10%	10 %
10000	N/A	3%	5%	5 %

Frame rate [fps]	Oqus 5+-series FOV	Oqus 5-series FOV	Oqus 7+-series FOV
50	Full resolution (2048x2048, 4194304 pixels)	Full resolution (2352x1728, 4064256 pixels)	Full resolution (4096x3072, 12582912 pixels)
100	Full	Full	Full
200	Full (179 Hz)	Full (180 Hz)	Full
250	72%	75%	Full (300 Hz)
500	36%	38%	60%
1000	18%	19%	30%
2000	9%	9%	15%
5000	4%	4%	6%
10000	2%	2%	3%

The following resolutions are available with the high speed sensor modes for the 2c-, 3+-, 5+- and 7+-series cameras.

Frame rate [fps]	Oqus 2c-series FOV (high speed mode)	Oqus 3+-series FOV (high speed mode)	Oqus 5+-series FOV (high speed mode)	Oqus 7+-series FOV (high speed mode)
50	Full resolution (1024x544, 557056 pixels)	Full resolution (648x512, 331776 pixels)	Full resolution (1024x1024, 1048576 pixels)	Full resolution (2048x1736, 3555328 pixels)
100	Full	Full	Full	Full
200	Full	Full	Full	Full
250	Full	Full	Full (355 Hz)	Full
500	Full (666 Hz)	Full	71%	Full
1000	67%	Full	36%	Full (1121 Hz)
2000	33%	Full (1764 Hz)	18%	55%
5000	13%	35%	7%	22%
10000	7%	18%	4%	11%

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: When using Oqus high-speed video, take into account prolonged data fetch times after a capture. The data transfer time for a full buffer memory (1.1 GB) can be several minutes.

Note: The 4 and 7+ series cameras are 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 - 14 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 and the recording time for images. The recording times for images are calculated for uncompressed images and VGA resolution.

The size of a marker frame in bytes is calculated through the following formula: 36 + 12 * "Number of Markers". However since the marker data is fetched as fast as possible during the capture the recording time cannot be easily calculated. In most setups the recording time is in fact infinite.

Memory size [MB]	Recording time - images	
	VGA @ 30 fps	VGA @ 500 fps*
144	16 s	1.0 s
288	32 s	2.0 s
576	65 s	3.9 s
1152	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 - 13 for information on how to change the strobe.

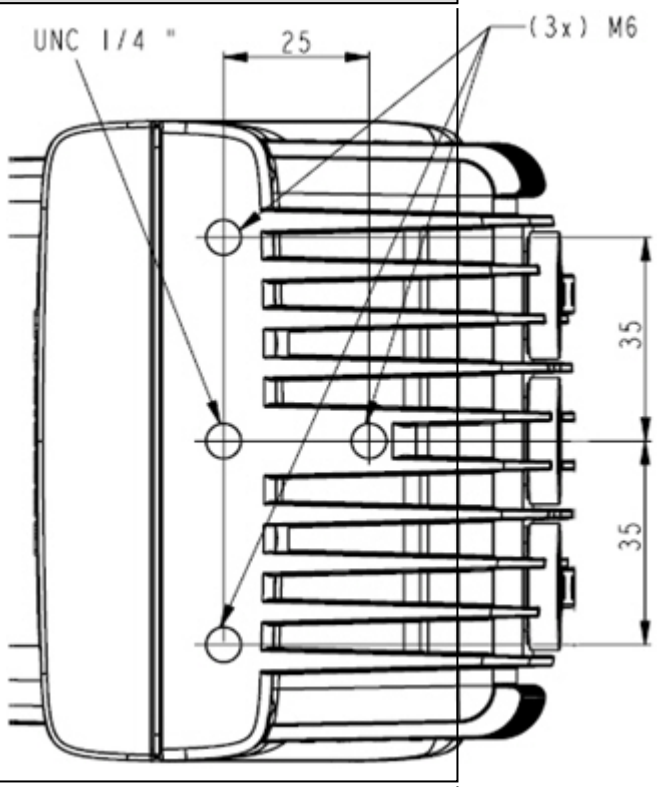
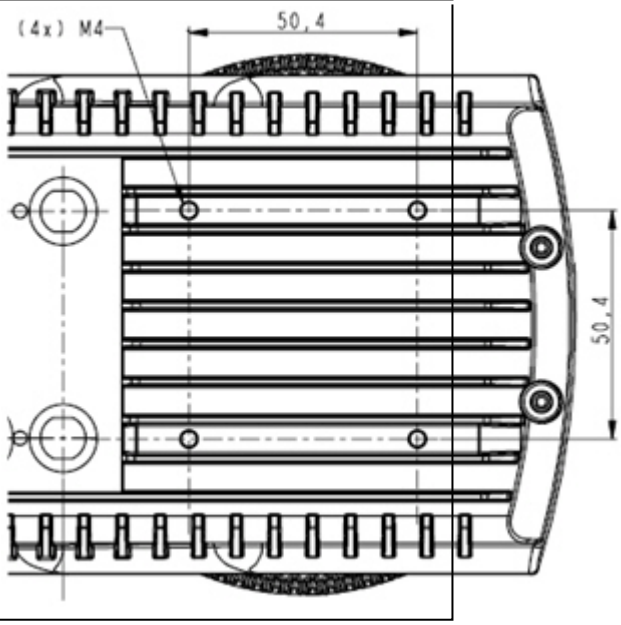
Environmental protection

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 (Oqus 6+/7+: 0-30 °C)
Position data noise level	+/- 1 camera units
Marker support	Both passive and active
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+/4/5/6+/7+, 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	2c-series (Video)	3-series/ 3+-series	4-series
CMOS sensor size (pixels)	640×480	1920×1088	1280×1024	1696x1710
Marker position resolution (sub pixels)	41000 x 30000	N/A	82000 x 65000	110000 x 110000
Maximum frame rate at full resolution and field-of-view	247 fps	N/A	502 fps	485 fps
Maximum frame rate at reduced field-of-view	1000 fps	N/A	10000 fps	10000 fps
Maximum video frame rate (full resolution)	247 fps	337 fps	502 fps	N/A
Maximum video buffer capacity (full resolution)	3800 frames	540 frames	900 frames	N/A
Maximum video buffer capacity (full resolution and frame rate)	15.2 s	1.6 s	1.8 s	N/A
In-camera MJPEG compression	No	Yes 24 fps, full resolution	No	No
Active filtering	Yes	Yes	Yes (only 3+)	Yes
High speed mode	No	Yes (1024x544 at 666 Hz) (Video only: 480*272 at 1300 Hz)	Yes (only 3+, 648x512 at 1764 Hz)	No

	5+-series	5-series	6+-series	7+-series
CMOS sensor size (pixels)	2048x2048	2352×1728	3072x1984	4096x3072
Marker position resolution (sub pixels)	130000 x 130000	150000 x 110000	200000 x 127000	260000 x 200000
Maximum frame rate at full resolution and field-of-view	179 fps	180 fps	450 fps	300 fps
Maximum frame rate at reduced field-of-view	10000 fps	10000 fps	10000 fps	10000 fps
Maximum video frame rate (full resolution)	179 fps	180 fps	N/A	N/A
Maximum video buffer capacity (full resolution)	280 frames	290 frames	N/A	N/A
Maximum video buffer capacity (full resolution and frame rate)	1.5 s	1.6 s	N/A	N/A
In-camera MJPEG compression	Yes 12 fps, full resolution	No	Yes 6 fps, full resolution	Yes 3 fps, full resolution
Active filtering	Yes	No	Yes	Yes
High speed mode	Yes (1024x1024 at 355 Hz) (Video only: 480*512 at 694 Hz and 240*256 at 1328 Hz)	No	Yes (1536x992 at 1660 Hz)	Yes (2048x1536 at 1121 Hz)

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” on the previous page.
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” on the previous page.

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.

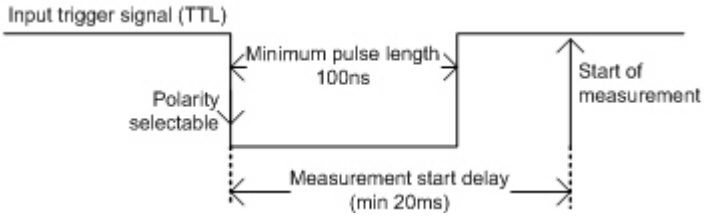
One Oqus camera in a system can be equipped with WLAN for wireless communication. Then the camera with WLAN communicates wireless with the host computer. The rest of the cameras are connected through Ethernet in a daisy-chain as usual.

Digital and analog I/O

The Oqus camera provides several inputs and outputs, both digital and analog.

Trigger input

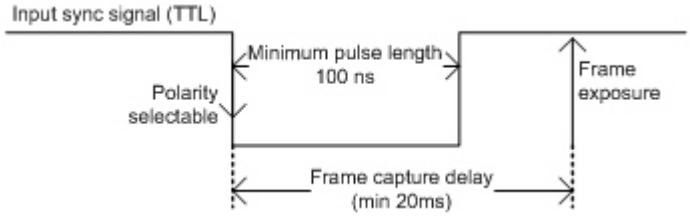
The camera can receive a trigger signal from an external TTL source. Depending on the selected mode, the external trigger can either start a pending measurement or switch from pre- to post trigger measurement. In a multi-camera environment, the trigger can be connected to any camera in the system. The time between the signal is asserted and the action in the camera can be selected on the **Synchronization** 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 **Synchronization** page in the **Project options** dialog.

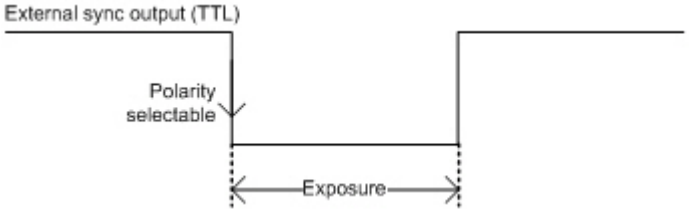


Synchronization output

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 **Synchronization** 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 - Extension	N/A	Standard
		p	IP67
		m	MRI
		u	Underwater
		c	Color sensor

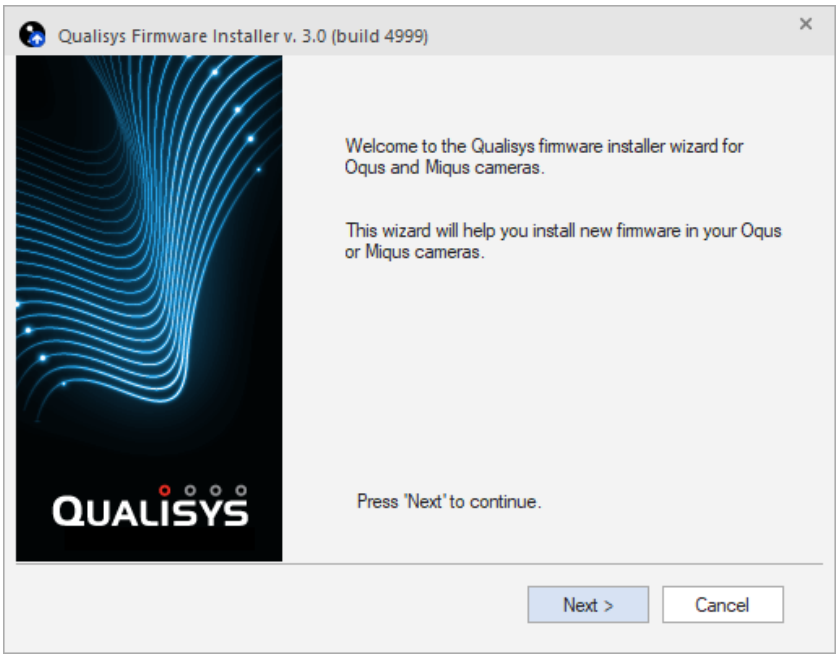
An Oqus camera can for example be called 310p, which is a Oqus with 1.3 Mpixel sensor, High-speed and IP67 protection.

Note: It will say PLUS under the number if the camera is a 3+-series camera.

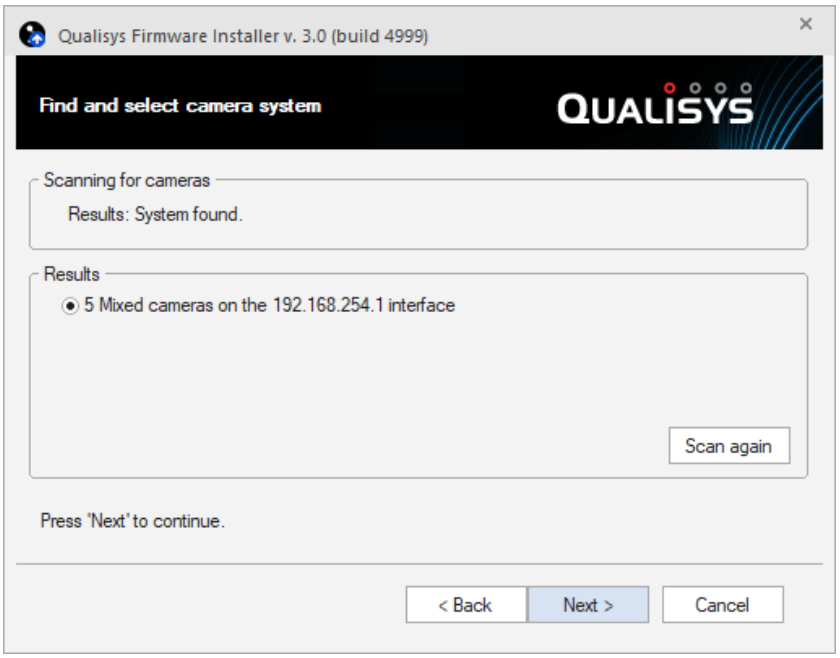
Appendix B: Qualisys Firmware Installer (QFI)

How to use Qualisys Firmware Installer (QFI)

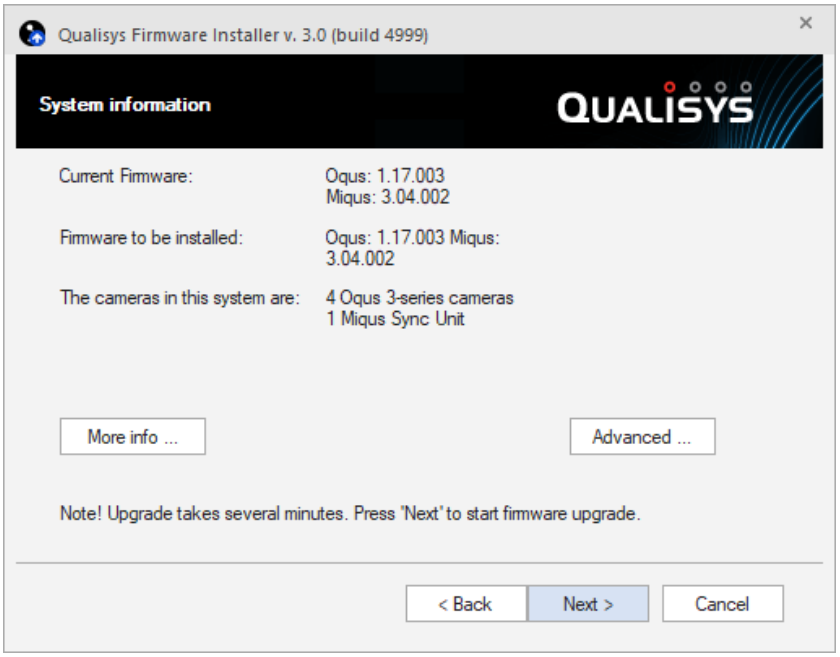
The Qualisys Firmware Installer (QFI) is used to install firmware on Qualisys cameras. QFI is organized as a Wizard. In a regular firmware upgrade just click **Next** until the upgrade is finished, an upgrade may take a few minutes to perform. The different steps are described below.



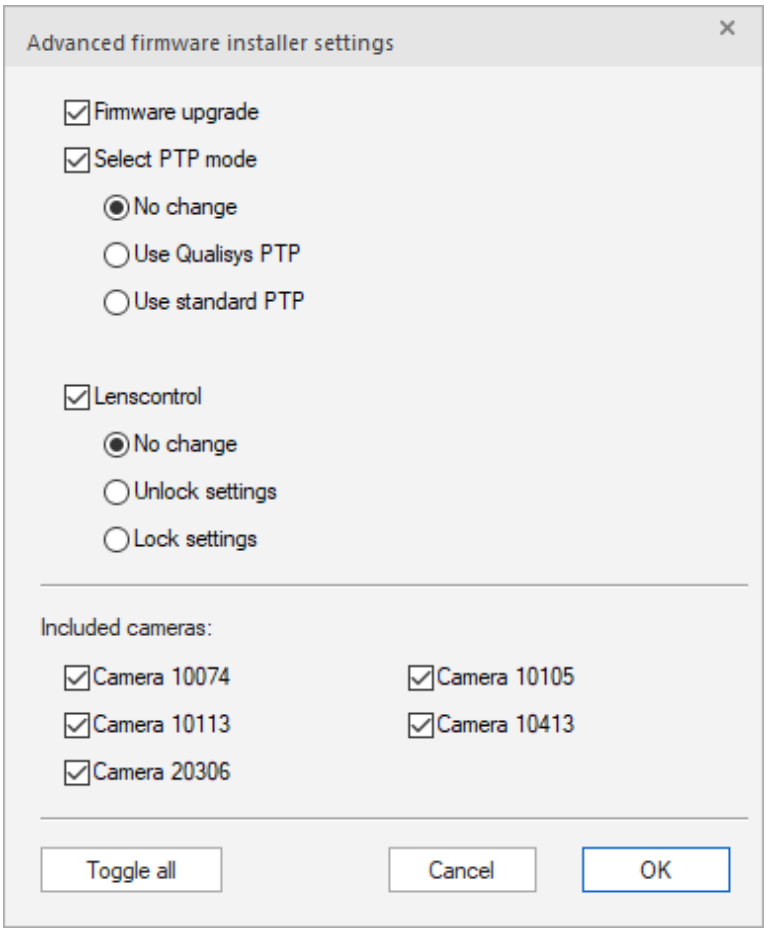
Click **Next** to start looking for Oqus cameras connected to the computer. The cameras must have started completely before you start locating the cameras.



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 modify **PTP Mode** or **Lens Control**.

PTP Mode
The PTP mode options are:

No change: Keep the current PTP mode. This is the default choice.

Use Qualisys PTP: This is the default PTP mode used for synchronization of the Qualisys devices.

Use standard PTP: This PTP mode needs to be selected for PTP synchronization of the Qualisys devices with an external clock master. For more information, see chapter “How to use PTP sync with an external clock master (Camera Sync Unit)” on page 480.

Lens Control

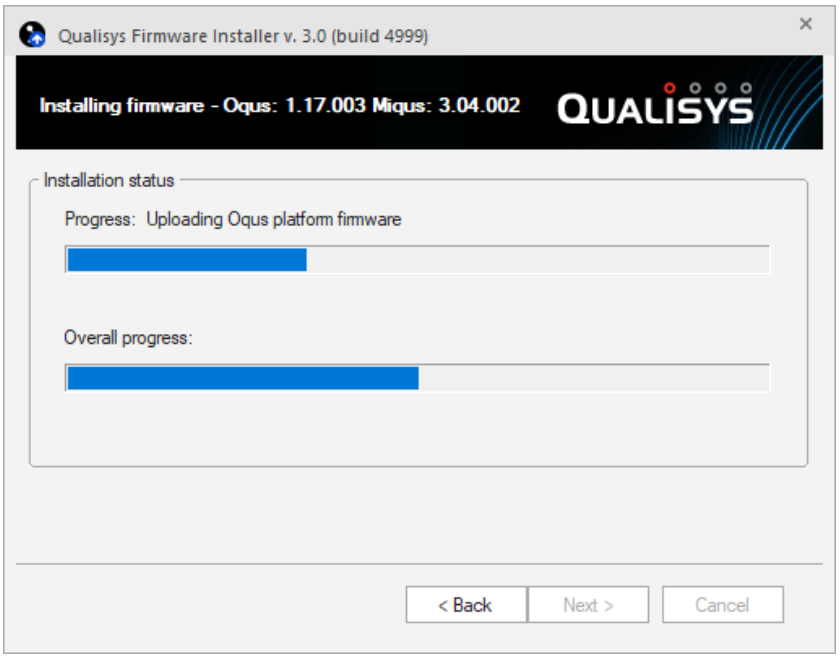
No change: Keep the current lens control mode. This is the default choice.

Unlock settings: Enable focus and aperture control from QTM for cameras with a motorized lens. This is the default mode.

Lock settings: Disable focus and aperture control from QTM. In this mode, the communication with the lens is disabled and the lens control parameters are no longer shown in QTM. This setting can be useful to fix the focus and

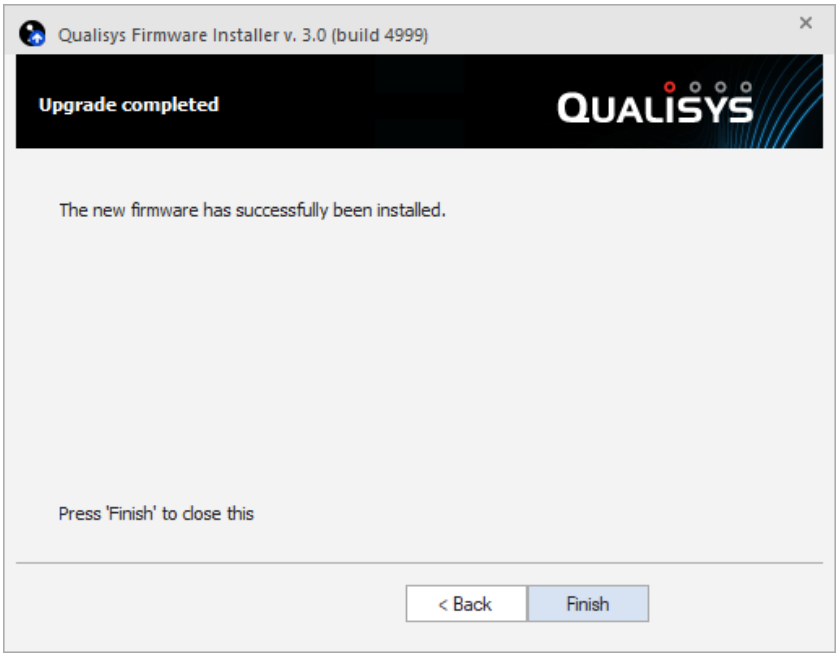
aperture settings once they have been set to their optimal values in a fixed camera setup.

Below these settings is 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



Click **Finish**.

Note: If QFI reports errors after the firmware upgrade, it is recommended to run QFI again.

Appendix C: Analog input/output hardware

Analog I/O 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 (Instacal).
   QTM supports the following analog boards:
   - USB-2533
   - USB-1608G
   - See also chapter “Old analog hardware” below for an overview of depreciated Analog I/O devices.
2. A synchronization cable.

Furthermore, QTM supports the PCI-6703 board for analog output, used for 6DOF analog output (see chapter “6DOF analog output” on page 346).

Normally the entire system is installed in the measurement computer before it is delivered by Qualisys AB. However, if you have a 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.

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.

USB-1616FS
The USB-1616FS is depreciated because of incompatible drivers on Windows 7 and later. It is no longer tested in QTM (last tested in QTM 2.10 on Window XP).

PCI-DAS6402/16
The PCI-DAS6402/16 is no longer tested in QTM (last tested in QTM 2.12).

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. Instacal is included in the QTM installer. Alternatively, the latest version of Instacal can always be downloaded from www.mccdaq.com.

After Instacal has been installed you can go on to install and configure your A/D board, see chapter “Installing the USB-2533 board” on the facing page.

For more information about InstaCal read the **README.TXT** file from Measurement Computing that is put in the folder for the Instacal software.

USB A/D board

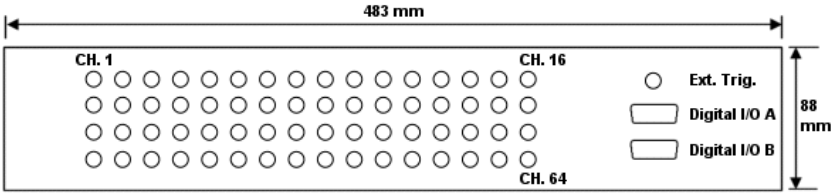
Qualisys currently supports the following analog data acquisition boards.

The 64 channel board USB-2533

The 16 channel board USB-1608G

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. For instructions how to install the board see chapter “Installing the USB-2533 board” on the facing page.

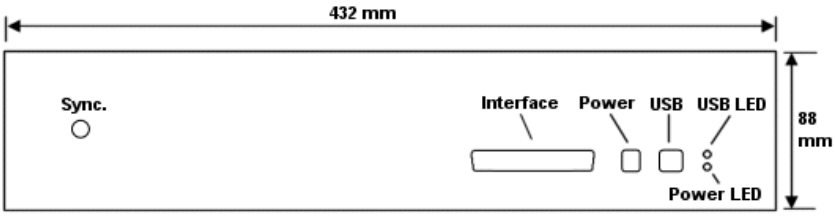


The following connections are available on the front view of the board:

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 C - 8.



The following connections are available on the rear view of the board:

SYNC
BNC connection for frame synchronization of the analog capture.

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.

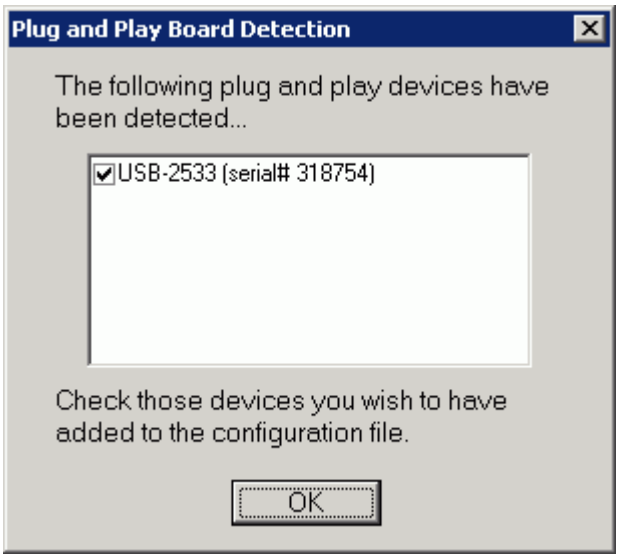
USB LED
The USB LED is lit when the board is connected to a computer.

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.

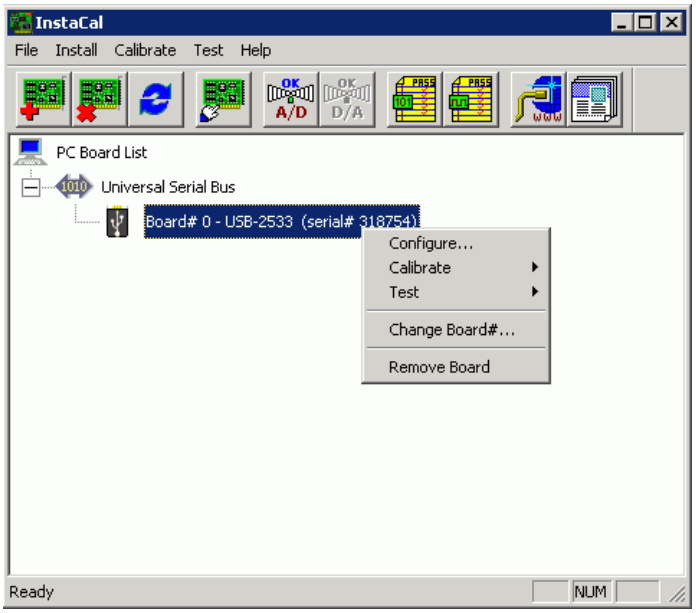
Installing the USB-2533 board

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.

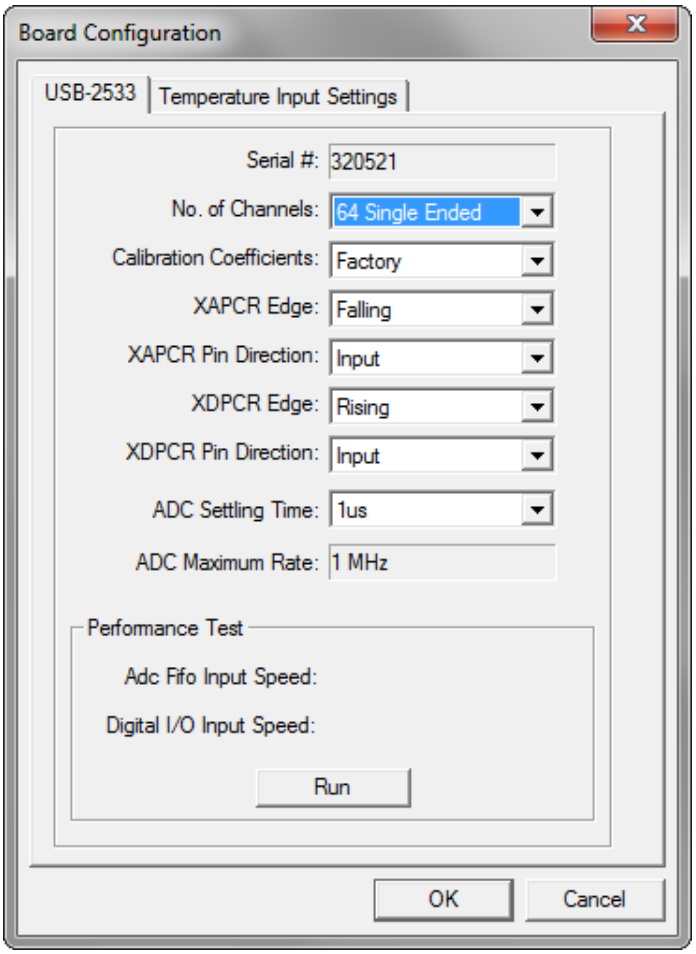


8. Right-click on the board and then click on **Configure**. Then change **No. of Channels** to **64 Single Ended**. The reason for using single ended is among other things that force plates usually have this type of signal output.

   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.

   It is important to check that the **XAPCR Edge** setting matches the setting for the Synchronization output on the **Synchronization** page in the **Project options** dialog. The default **TTL signal polarity** is **Negative** which matches **XAPCR Edge** = **Falling**.

   Warning: Keep the **XAPCR Pin Direction** setting to **Input**. Changing this setting may cause damage to the board or to the camera.

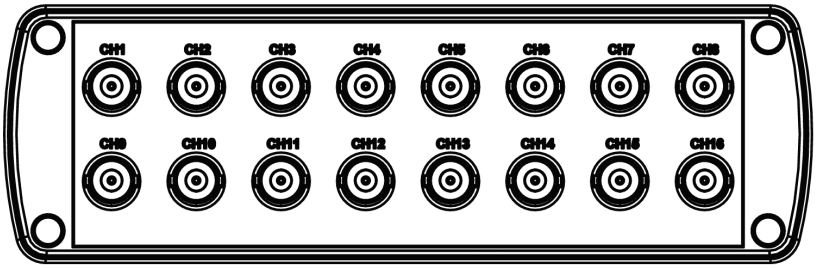


9. Click **OK** and then exit Instacal.

When the board is properly installed it will be listed on the **Input devices** page in the **Project options** dialog in QTM.

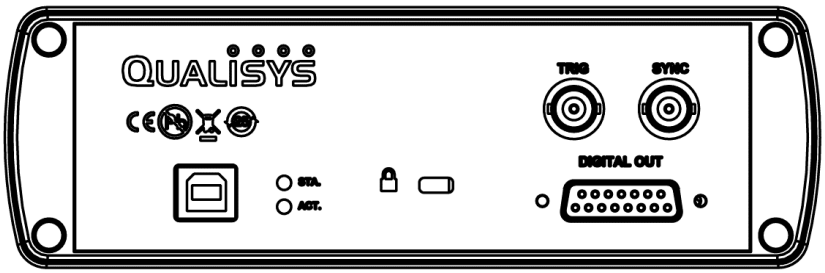
USB-1608G

The USB-1608G is a portable A/D board that can easily be connected to any computer with a 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. For instructions how to install the board see chapter “Installing the USB-1608G board” on the next page.



The following connections are available on the front view of the board:

CH1 - CH16
BNC connections for the 16 analog channels.



The following connections are available on the rear view of the board:

SYNC
BNC connection for frame synchronization of the analog capture.

TRIG
BNC connection for synchronous start of the analog capture, see chapter "Hardware synchronization" on page C - 8.

DIGITAL OUT
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 is then the digital I/O and pin 13-15 is ground.

USB
USB connection to the measurement computer. The cable is supplied by Qualisys.

STA. LED
The Status LED turns on when the device is detected and installed on the computer.

ACT. LED
The Activity LED blinks when data is transferred, and is off otherwise.

USB-1608G specifications

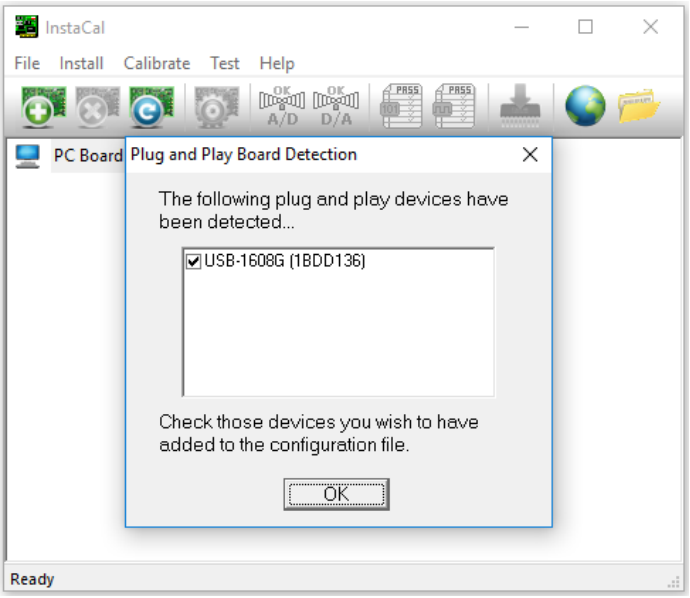
Materials	Extruded and die-cast Aluminum housing
Weight	910g
Channels	16 single ended
ADC resolution	16 bits
Input voltage range	±10V, ±5V, ±2V and ±1V software selectable per channel
Trigger input	5V TTL
Synchronization input	5V TTL
Digital outputs	12 5V TTL
Data/power connection	Compatible with both USB 1.1 and USB 2.0 ports

Installing the USB-1608G board

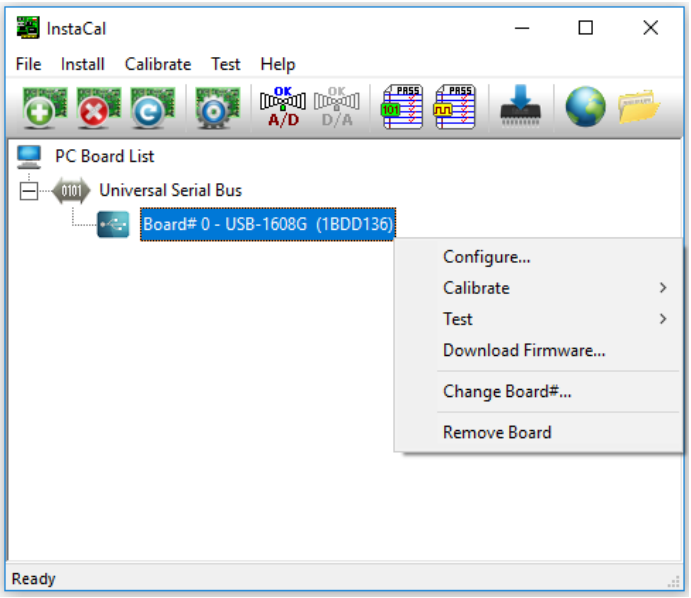
Installing the USB-1608G board requires that InstaCal has been installed on the computer. If needed, reinstall QTM with the InstaCal option checked. Follow this procedure to install the USB A/D board:

1. Connect the USB-1608G board to USB port.
2. Start InstaCal via the Windows start menu.

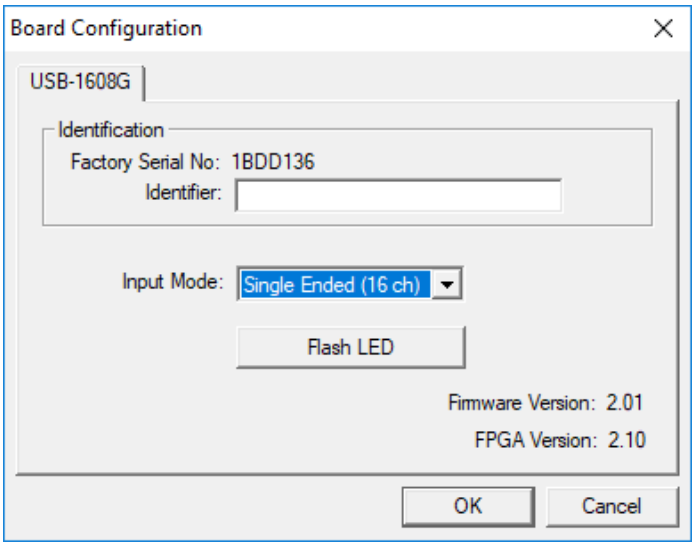
3. The USB-1608G board should appear in the plug and play dialog. Click OK.



4. Open the configuration dialog (double click or right click and select "Configure").



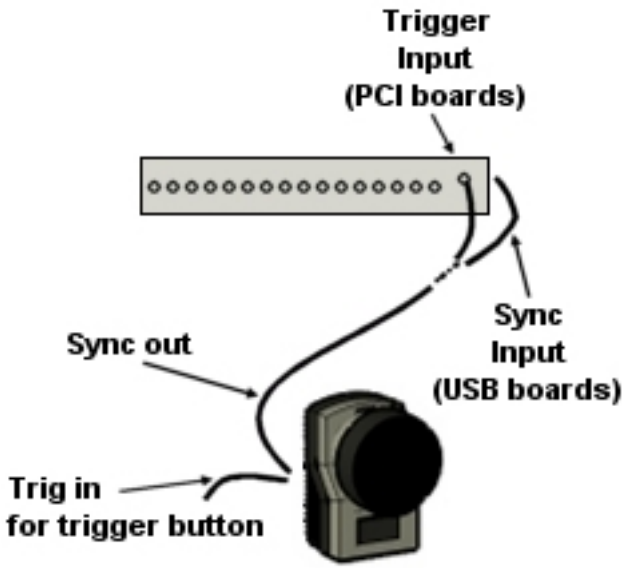
5. Set "Input mode" to "Single Ended (16 ch).



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.



USB-2533/USB-1608G

The recommended synchronization option is the frame synchronization, see chapter “Sample rate” on page 167. For this option you must connect one of the Sync outputs (**Out 1** or **Out 2**) of the Camera Sync Unit or the **Sync out** connection from one of the cameras (Oqus) to the **Sync** connection on the back of the A/D board.

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. To use this option connect one of the Sync outputs (**Out 1**, **Out 2** or **Meas. time**) of the Camera Sync Unit or

the **Sync out** connection from one of the cameras (Oqus) to the **External trigger** or **Trig** connection on the A/D board.

Note: The analog boards come installed in a box with BNC connections when purchased via Qualisys AB.

Appendix D: 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, Mega and Delsys EMG systems are integrated in QTM so that they can be collected directly without an extra analog board. The chapters “Noraxon EMG” below, “Mega EMG” on page D - 17 and “Delsys Trigno EMG” on page D - 26 describe the settings and configurations for the use of these systems in QTM.

Noraxon EMG

QTM supports integration with the Noraxon TeleMyo 2400T G2 EMG, TeleMyo DTS Belt Receiver, DTS Desk Receiver and Ultium. The TeleMyo 2400T G2 has wireless communication and 4 to 16 channels. On the Noraxon TeleMyo DTS and Ultium systems each EMG channel consists of a wireless device. A wide range of sensors can be connected to the EMGs.

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 Noraxon hardware 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 **Noraxon** under **Input Devices**, **EMGs** in the **Project options** dialog, run the installer again and make sure to include the Noraxon files.
2. Connect the Noraxon device according to the description in chapters “Noraxon computer connection” on the next page and “Noraxon trigger connection” on the next page.
3. Start the Noraxon EMG and wait for the communication to be up and running.
4. Go to the **Noraxon** page under **EMGs** in the **Project options** dialog. Click on **Device Setup** to run the Noraxon hardware setup.
5. Remove any unused devices (e.g. the player devices)
6. Click insert devices, select your device and click OK.

7. Press the Detect button to retrieve the hardware settings.
8. For configuring your hardware setup, follow the instructions for your device under “Hardware setup” on page D - 10.

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 Belt Receiver.

Directly to the TeleMyo DTS Belt Receiver, DTS Desk Receiver or Ultium
It is recommended to connect the Belt Receiver directly to the computer even though it can be used with the other receivers above.

They are all connected via USB and use the driver nxnusb 1.35 from Noraxon. It can be downloaded from http://www.noraxon.com/support/downloads/ (choose "Drivers" and download "Noraxon USB driver"). Unzip the driver and place it where you can find it. When the Noraxon EMG is connected the first time Windows will ask for the driver, make sure that you browse to the correct folder.

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 four interfaces, see instructions in the chapters below.

Trigger connection to Ultium

The trigger is connected directly to the Ultium device, follow the steps below.

1. Split the trigger button signal with a T-coupling and then connect the signal to the **Sync** connector on the Ultium device.
2. In the **Noraxon Hardware Setup**, make sure to check the **Invert Sync** option, see chapter “Hardware setup - Ultium” on page D - 11. In QTM on the **Synchronization** page of the project options, make sure that TTL signal edge of the used trigger port is set to **Negative**.

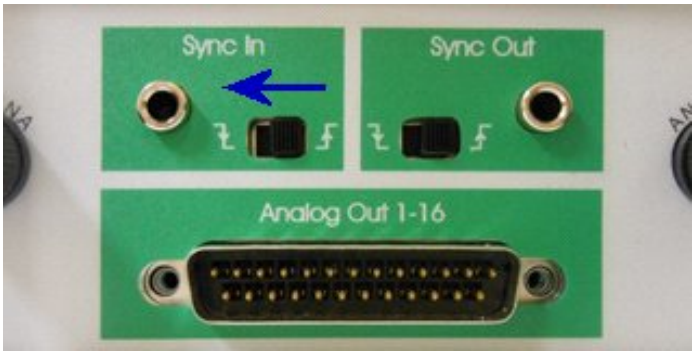
   IMPORTANT: A wrong setting of the **Invert Sync** option will lead to synchronization errors.
3. You will also need to assign the sensors, for instructions see chapter “Noraxon QTM settings” on page D - 9.
4. Test the system by making a measurement in QTM, see chapter “Making a measurement with Noraxon”.

Trigger connection to DTS Desk Receiver

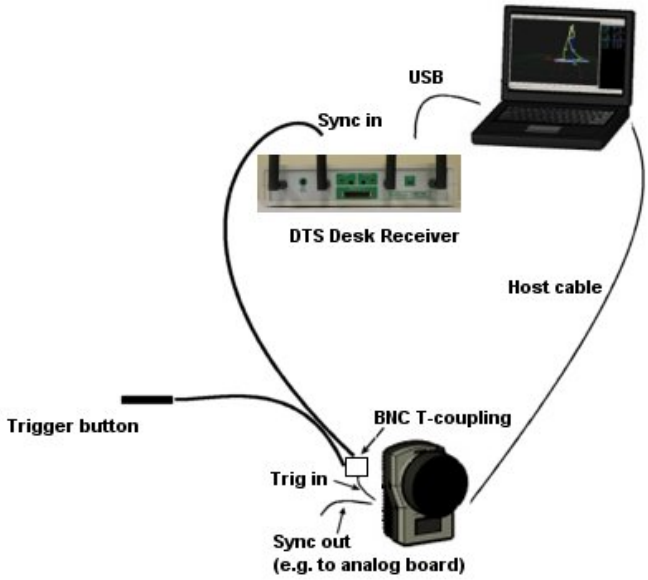
The trigger is connected directly to the DTS Desk Receiver, follow the steps below.

1. Split the trigger button signal with a T-coupling and then connect the signal to the **Sync in** input on the DTS Desk Receiver.

⚠ **IMPORTANT:** On the DTS desk receiver, make sure that you set the polarity of the input to falling edge, i.e. move the switch to the left side. In QTM on the **Synchronization** page of the project options, make sure that TTL signal edge of the used trigger port is set to **Negative**.



2. You will also need to assign the sensors, for instructions see chapter "Noraxon QTM settings" on page D - 9.
3. Test the system by making a measurement in QTM, see chapter "Making a measurement with Noraxon".

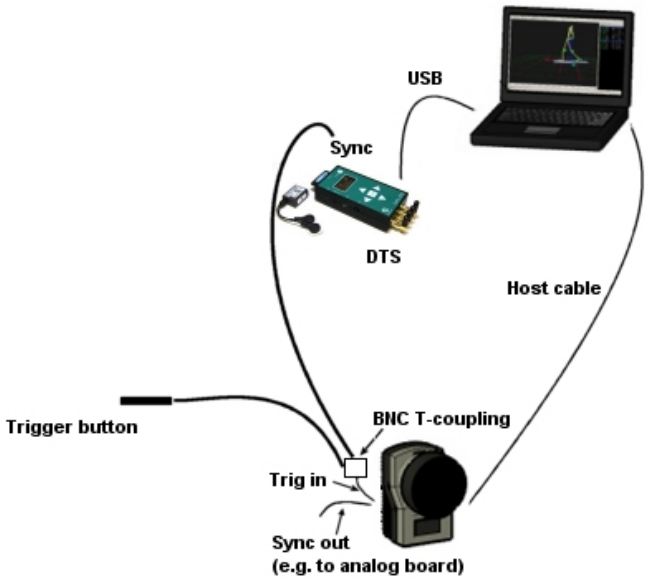


Trigger connection to TeleMyo DTS Belt Receiver

The recommended trigger connection for the TeleMyo DTS Belt Receiver 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, make sure that the **Invert** switch is set to **Yes**.

2. You will also need to assign the sensors both in QTM and on the receiver, for instructions see chapter “Hardware setup” on page D - 10 respectively the Noraxon manual.
3. Test the system by making a measurement in QTM, see chapter “Making a measurement with Noraxon”.



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.

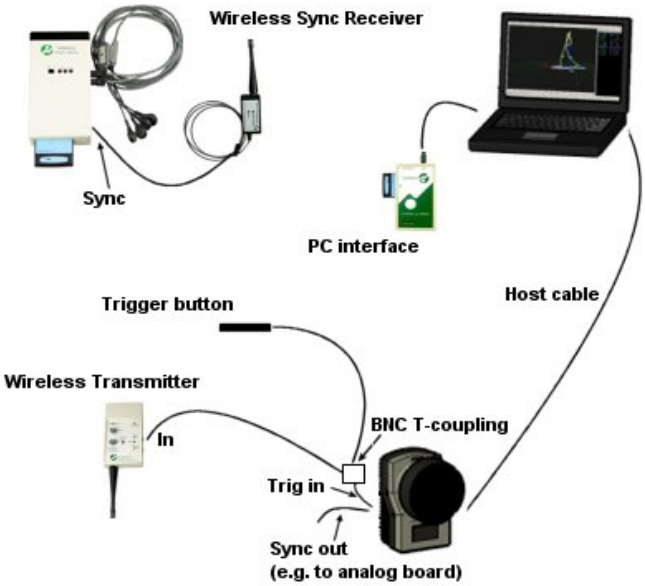
1. Do NOT turn on the Wireless Transmitter before the last step.
2. Connect the Wireless Sync Receiver to the **Sync in** port on the Noraxon system.
3. Connect the Wireless Transmitter to the **External trigger input** connector on the **Control** port cable of the camera system. Use the wire with a 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 **Synchronization** page in the **Project options** dialog.

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.



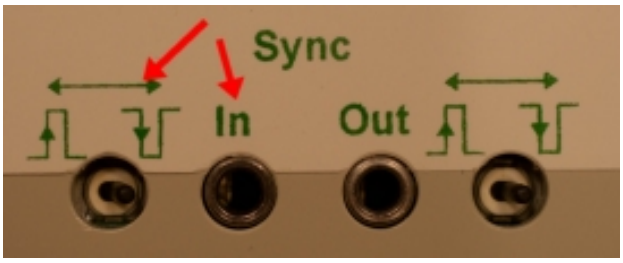
Trigger connection to TeleMyo 2400R G2 Mini Receiver or TeleMyo 2400R G2 Analog Output Receiver

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**.

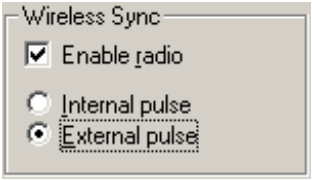
   a. Set the settings on the Receiver interface to negative edge for the **In** connection.

   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.

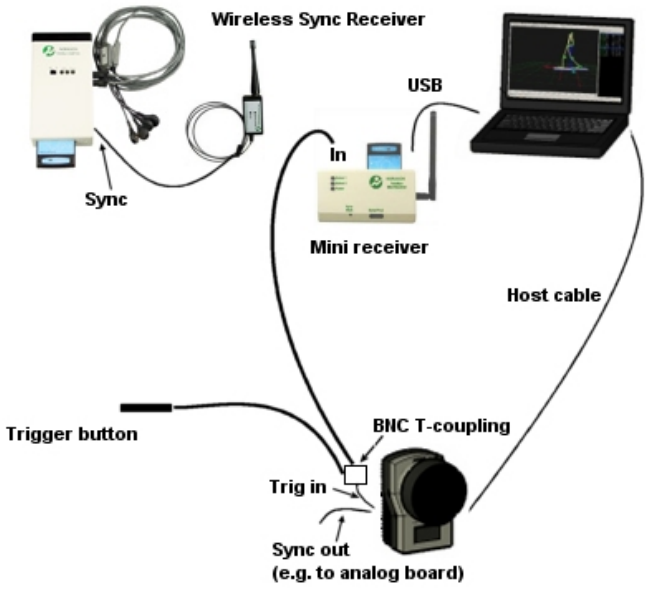
   c. The camera must be set to trigger on negative edge, which is done on the **Synchronization**page in the **Project options** dialog.

3. To activate the trigger input on the Receiver interface you must go to the **USB receiver configuration dialog** in QTM, see chapter “Hardware setup” on page D - 10. 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.

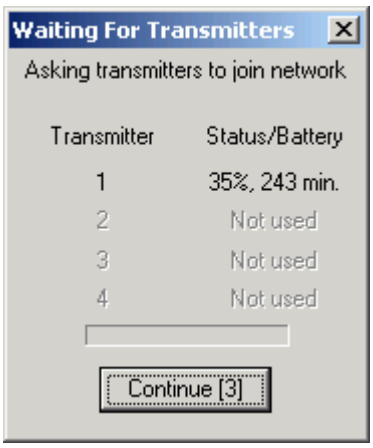


Making a measurement with Noraxon

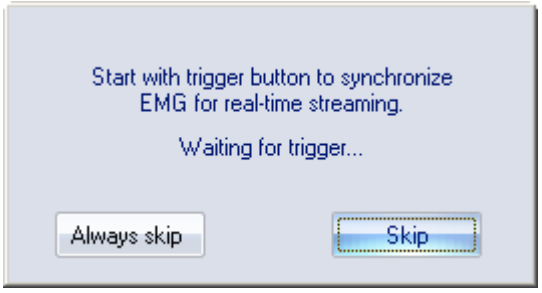
With the Noraxon EMG it is possible to capture up to 16 EMG channels or other type of sensor data, depending on the model. Follow these steps to make a measurement with Noraxon EMG in QTM.

1. The first time a Noraxon EMG is connected to a computer you must follow the instructions in the chapter “Noraxon installation” on page D - 1.
2. Before starting a new measurement connect the synchronization devices according to the description in chapter “Noraxon trigger connection” on page D - 2.
3. Open the **Project options** dialog and go to the **Synchronization** page. Set the camera system to start on external trigger.
4. Go to the **Noraxon** page under **Input Devices** > **EMGs** and select the settings below, for more information see chapter “Noraxon QTM settings” on page D - 9.
   a. 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.
   b. 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.
   c. 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.
5. 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

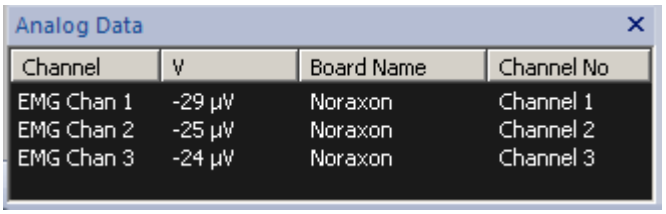
while in preview and when opening the **Start capture** dialog make sure that you never click **Cancel** in the dialog.



6. 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.



7. Open the **Data info window** on the **View** menu and then right-click and select **Display Analog data**.
8. 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.



9. When starting a capture the start of the EMG data will be synchronized with the motion data by the external trigger signal.

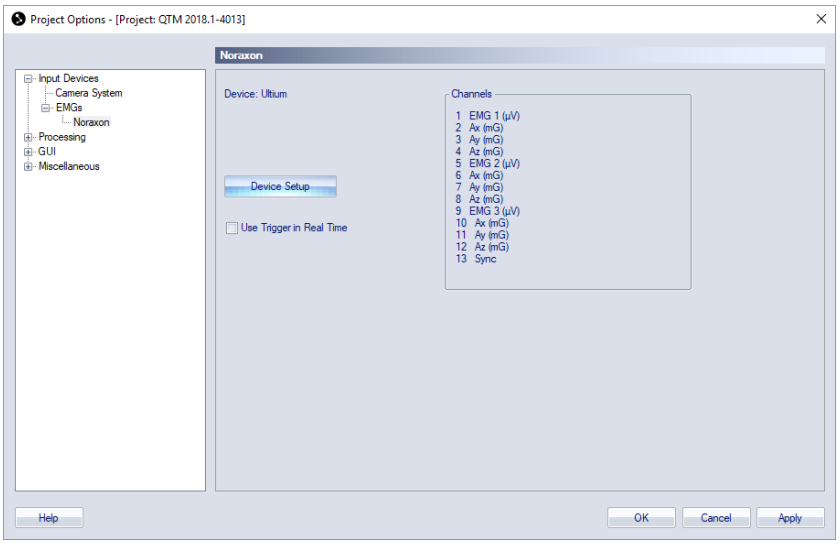
Export Noraxon EMG data

When the EMG data is exported to C3D format 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.

For MAT and TSV export the EMG data and accelerometer data is exported separately with the original capture frequency.

Noraxon QTM settings

The **Noraxon EMG** page, with settings for the Noraxon EMG, is included in the **Project options** dialog when the additional files for Noraxon have been installed using the QTM installer.



The page contains the different options for the Noraxon EMG, depending on whether it is DTS or a TeleMyo 2400T G2. For the TeleMyo 2400T G2 you will also select the channels that are used on the page, see below.
To change any Noraxon settings that are not included on the page, click on the **Device Setup** option to open the Noraxon **Hardware setup** dialog. For more information see chapter “Hardware setup” on the next page.

To activate the Noraxon EMG you must enable it on the **Input Devices** page. 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.

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.

Show Sync Channel
Active this option to display the sync channel in the **Data info** window. It can be used for checking the synchronization between the EMG and the rest of the system, but also if you want to check that the sync signal is detected by Noraxon.

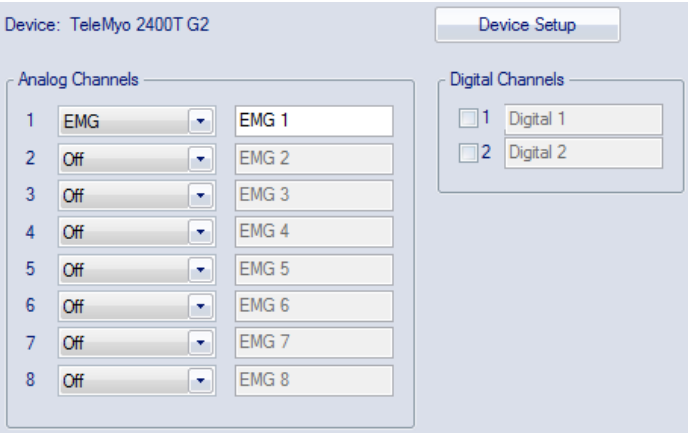
The following Noraxon settings from are displayed on the page:

Device
The type of Noraxon EMG that is used.

Channels

DTS systems
Displays the channels that have been activated for the Noraxon in the

Hardware setup dialog. This includes the sync channel which is activated with the **Show Sync Channel** option. If you want to change the channels you need to open the **Hardware setup** dialog with the **Device Setup** option.



TeleMyo 2400T G2
Select the type of sensor that is connect and enter a name fpr the channels that you want to use. There is also a **Digital Channels** box where you can activate the digital channels. Do not activate a channel that you do not use as it will only result in a larger file.

If you want to see any other settings click on the **Device Setup** option.

Hardware setup

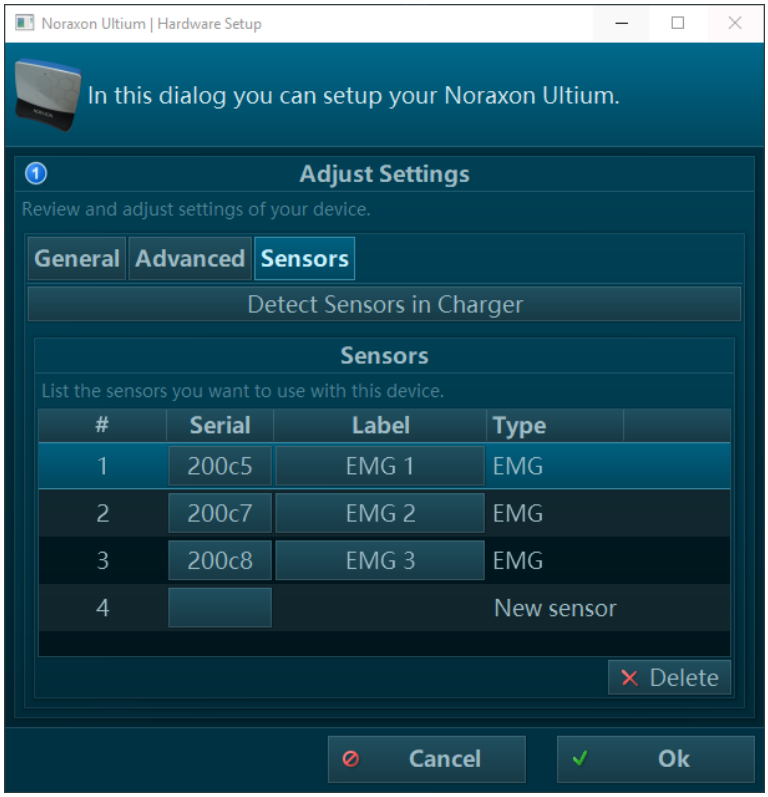
The **Hardware setup** dialog contains all of the settings needed to operate the Noraxon EMG. The descriptions below are only for the basic settings, for more information about the dialog refer to the Noraxon manuals.

The first time you open the dialog you will have to select your hardware. The connected device should be automatically detected. Make sure to only select the used device. The Player devices should be deleted or deselected.

When the hardware has been detected it is displayed in the **Hardware setup** dialog. Double-click on the device to open its settings. The settings depend on the used device, see respective section below.

Hardware setup - Ultium

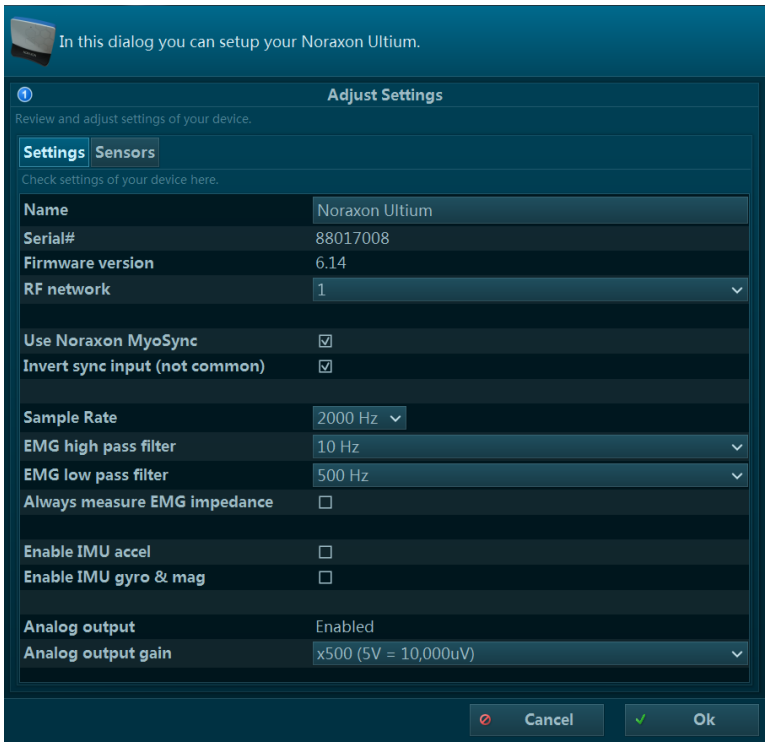
The hardware setup for the Ultium device contains settings both for the channels that are used, synchronization and other settings for the EMG.



The **Sensors** tab contains a list of the **Sensors** that are activated. You can add sensors automatically using the **Detect Sensors in Charger** button, or manually add sensors by filling in their serial number, which can be found on the sensor. The **Label** will be used for that sensor in the **Data info** window and exports of QTM. Do not activate a channel that you do not use as it will only result in a larger file.

The EMG data is displayed and exported in µV. For other types of sensors the data will still be displayed in the SI unit of that sensor. Except for the accelerometer which is displayed in gravitational units.

In the case of the **Accelerometer** the actual frequency of the data is only a third, because the x, y and z data is sent in sequence on the channel. Check the manual from Noraxon to see which other sensors that have the same behavior.



The **Settings** tab contains the hardware settings for the Noraxon EMG.

The following sync settings are required for a correct synchronization with the camera system.

Use Noraxon MyoSync
Check this option to allow that QTM triggers the EMG system correctly.

Invert Sync
Check this option for correct triggering. In QTM on the **Synchronization** page of the project options, make sure that TTL signal edge of the used trigger port is set to **Negative**.

For the remaining options, please refer to the Noraxon documentation.

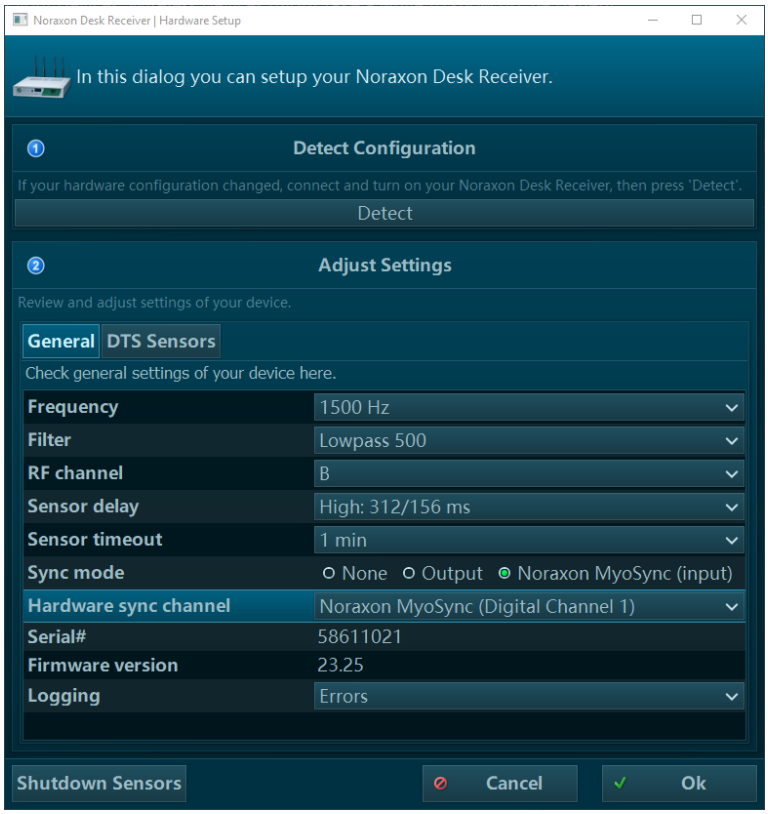
Hardware setup - DTS Desk Receiver

The hardware setup for the DTS Desk Receiver contains settings both for the channels that are used, synchronization and other settings for the EMG.

The **DTS Sensors** tab contains a list of the **Sensors** and **Foot Switches** that are activated. Add the number of sensors that you want to use in the list and assign the correct **Type** and enter the correct **Serial** number, which can be found on the sensor. The **Label** will be used for that sensor in the **Data info** window and exports of QTM. Do not activate a channel that you do not use as it will only result in a larger file.

The EMG data is displayed and exported in µV. For other types of sensors the data will still be displayed in the SI unit of that sensor. Except for the accelerometer which is displayed in g.

In the case of the **Accelerometer** the actual frequency of the data is only a third, because the x, y and z data is sent in sequence on the channel. Check the manual from Noraxon to see which other sensors that have the same behavior.



The **General** tab contains the hardware settings for the Noraxon EMG.

The following sync settings are required for a correct synchronization with the camera system.

Sync mode = Noraxon MyoSync (input)
Important setting to allow that QTM triggers the EMG system correctly.

Hardware sync input = Noraxon MyoSync (Digital Channel 1)
Important for correct operation of sync.

The following settings are the recommended for a DTS system, but you can change them if it is needed for the measurement.

Frequency = 1500 Hz
Suitable for all surface EMG recordings and other biomechanical sensors.

Filter = 500 Hz
Low pass filter applied to surface EMG sensors.

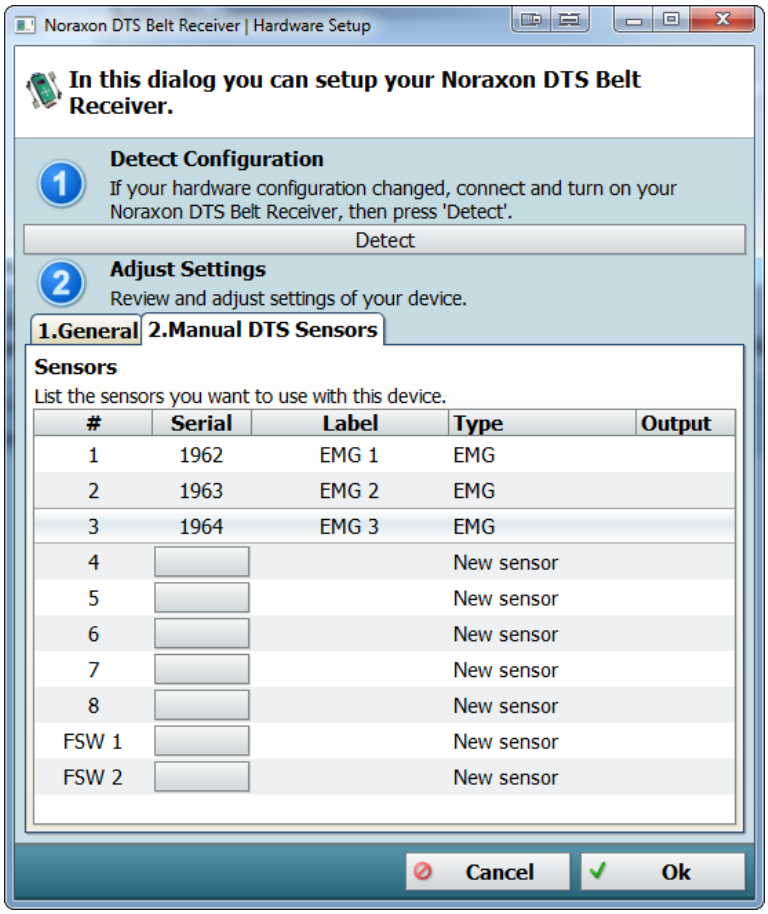
RF channel = B to G
The Wifi channel used for signal transmission. If the WiFi conditions are bad, it may be needed to try another channel (see Desk receiver hardware manual for more details).

Sensor delay = high 316/156 ms
The higher the delay the more stable is the signal transmission.

Sensor time out = 1 min
During this interval any transmission probe can rejoin the recording in case it left the transmission range.

Hardware setup - DTS Belt Receiver

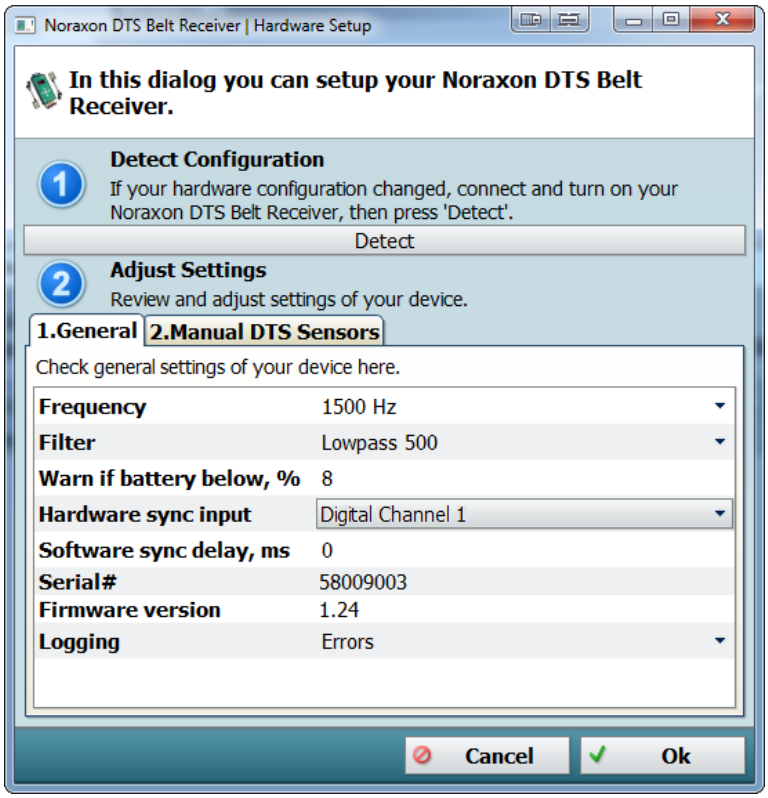
The hardware setup for the DTS Belt Receiver contains settings both for the channels that are used and general settings for the EMG.



The **Manual DTS Sensors** tab contains a list of the sensors that are used. Before you can add any sensors to the list you have to assign them in the Belt Receiver hardware (see Noraxon documentation for how to do this). Add the number of sensors that you want to use in the list and assign the correct **Type** and enter the correct **Serial** number, which can be found on the sensor. The **Label** will be used for that sensor in the **Data info** window and exports of QTM. Do not activate a channel that you do not use as it will only result in a larger file.

The EMG data is displayed and exported in µV. For other types of sensors the data will still be displayed in the SI unit of that sensor. Except for the accelerometer which is displayed in g.

In the case of the **Accelerometer** the actual frequency of the data is only a third, because the x, y and z data is sent in sequence on the channel. Check the manual from Noraxon to see which other sensors that have the same behavior.



The **General** tab contains the hardware settings for the Noraxon EMG.

The following sync settings are required for a correct synchronization with the camera system.

Software sync delay, ms = 0
This is needed for correct sync operation.

Hardware sync input = Noraxon MyoSync (Digital Channel 1)
Important for correct operation of sync.

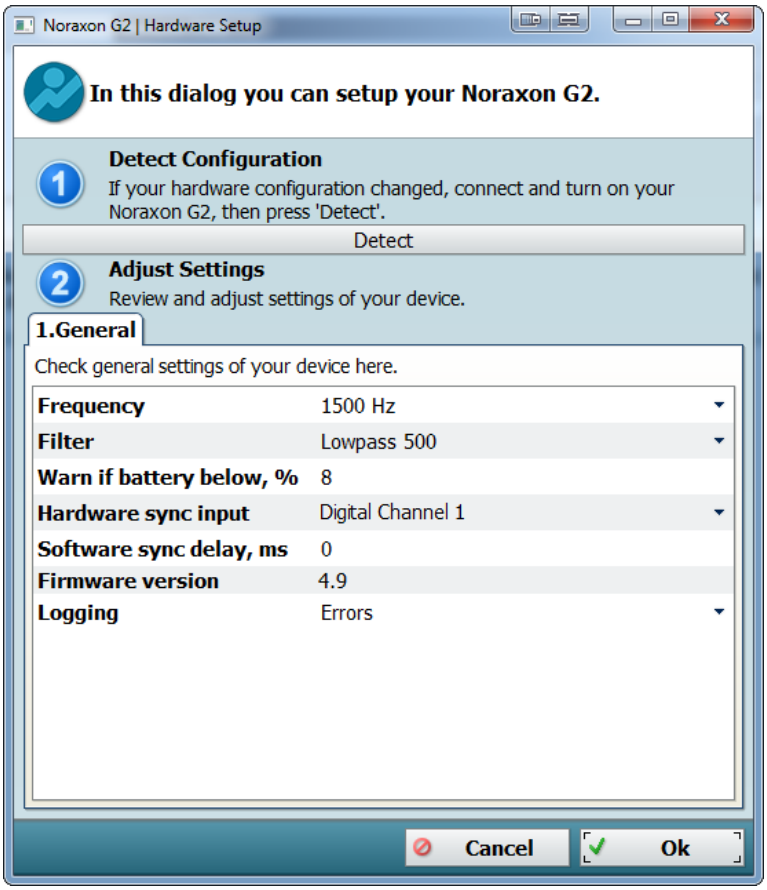
The following settings are the recommended for a DTS system, but you can change them if it is needed for the measurement.

Frequency = 1500 Hz
Suitable for all surface EMG recordings and other biomechanical sensors.

Filter = 500 Hz
Low pass filter applied to surface EMG sensors.

Hardware setup - TeleMyo 2400T G2

The hardware setup for the TeleMyo 2400T G2 contains general settings for the EMG.



The **General** tab contains the hardware settings for the Noraxon EMG.

The following sync settings are required for a correct synchronization with the camera system.

> **Software sync delay, ms** = 0
> This is needed for correct sync operation.
>
> **Hardware sync input** = Digital 1
> Important for correct operation of sync.

The following settings are the recommended for a DTS system, but you can change them if it is needed for the measurement.

> **Frequency** = 1500 Hz
> Suitable for all surface EMG recordings and other biomechanical sensors.
>
> **Filter** = 500 Hz
> Low pass filter applied to surface EMG sensors.

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 ME6000 must have firmware version 700059 or higher. Note that not all firmware versions allow to connect the device via WLAN. Refer to the MEGA manual or support for more information.

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 ME6000 must have firmware version 700059 1.5.3 or 700060 1.5.3, contact Mega at mega@megaemg.com if you need to update.

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 D - 20.

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.

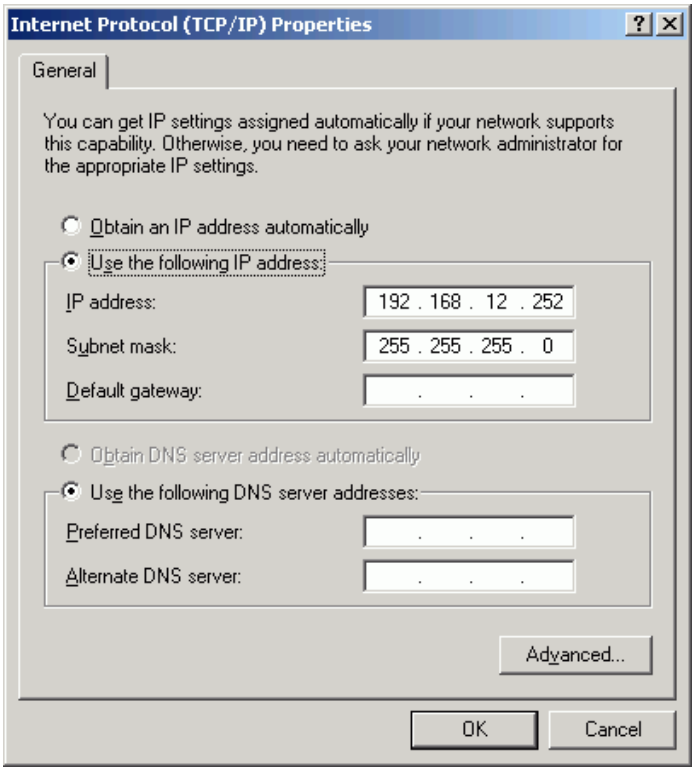
   Data encryption
   Not activated

   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)**.



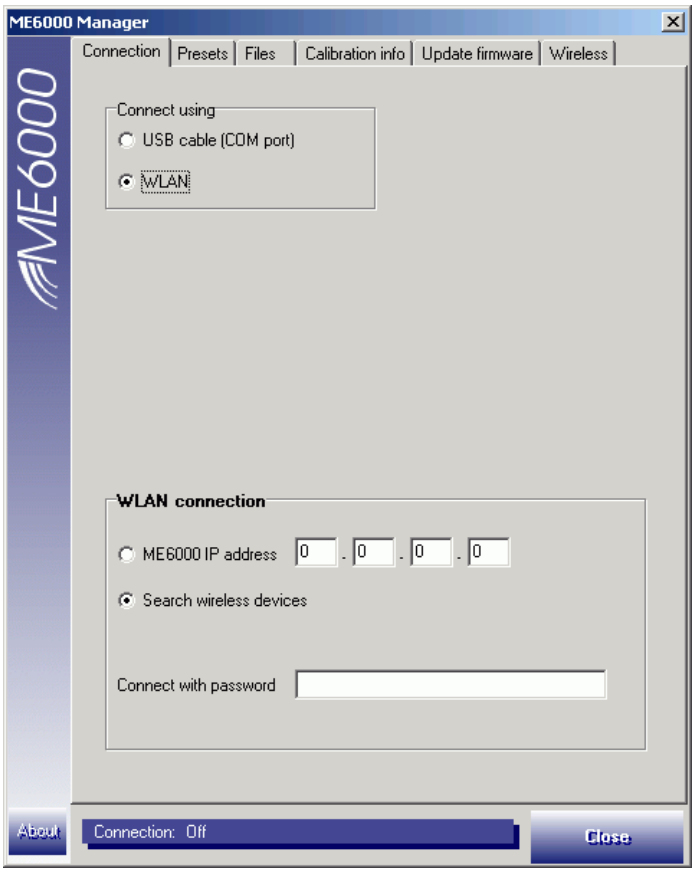
7. Change the IP setting to **Use the following IP address**.
   a. Set the same net as in **IP Parameters** on the Mega EMG. That means that the first three numbers must be the same and that the last number must be different from the Mega EMG and its router.
   b. Set the **Subnet mask** to 255.255.255.0.
   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

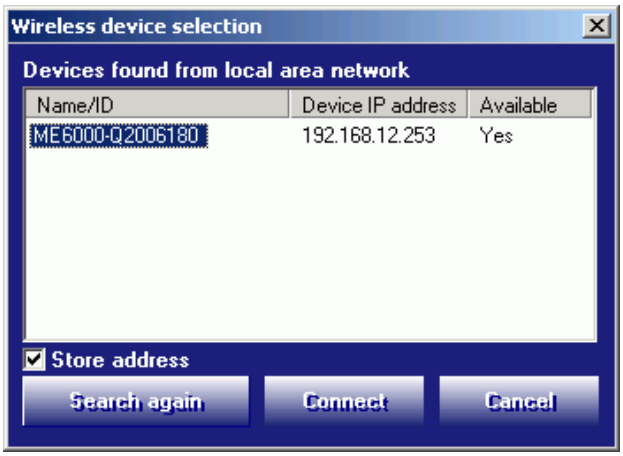
When the WLAN has been configured the next step is to connect to the Mega EMG in QTM.

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.

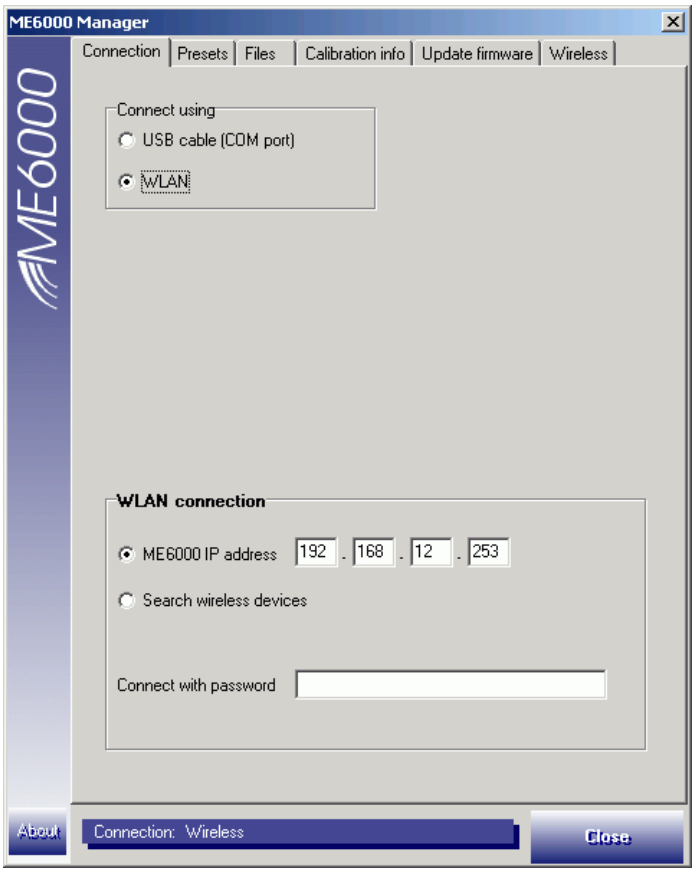


5. Click on any other tab to open the **Wireless device selection** dialog. If the EMG is not found click on **Search again**.
   a. When the device is found, select it so that the text turns blue.
   b. Then select the option **Store address** so that the IP address is saved to the next session. Otherwise 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.

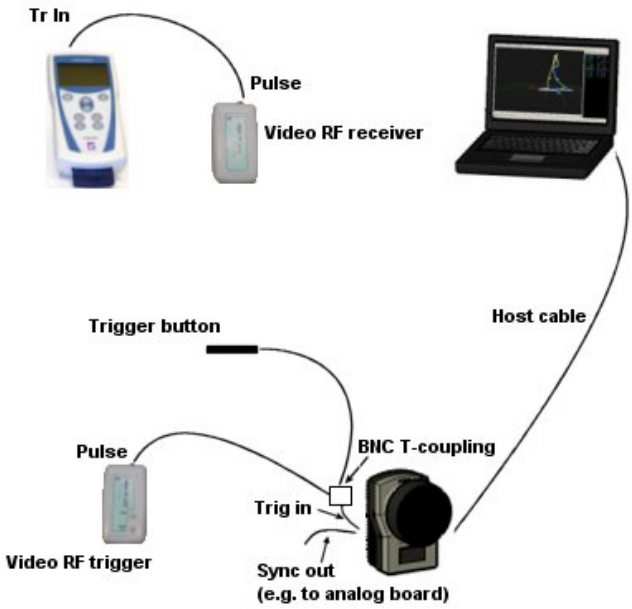


Mega hardware connections

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.
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.

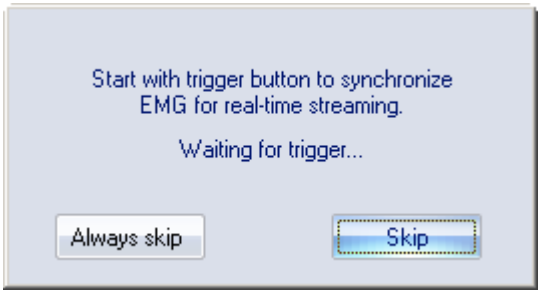


Making a measurement with Mega

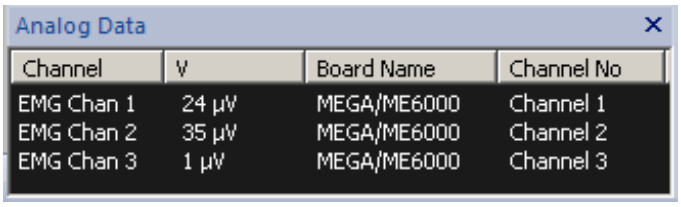
With the Mega ME6000 it is possible to capture up to 16 EMG channels or other type of sensor data, depending on the ME6000 model. Follow these steps to make a measurement with Mega EMG in QTM.

1. The first time a ME6000 is connected to a computer you must follow the instructions in the chapter “Mega installation” on page D - 18.
2. Before starting a new measurement connect the Video RF devices according to the description in chapter “Mega hardware connections” on the 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 **Synchronization** 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 D - 24.
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.

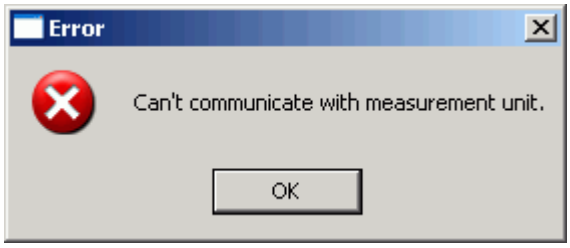


Channel	V	Board Name	Channel No
EMG Chan 1	24 µV	MEGA/ME6000	Channel 1
EMG Chan 2	35 µV	MEGA/ME6000	Channel 2
EMG Chan 3	1 µV	MEGA/ME6000	Channel 3

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 D - 18. 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.

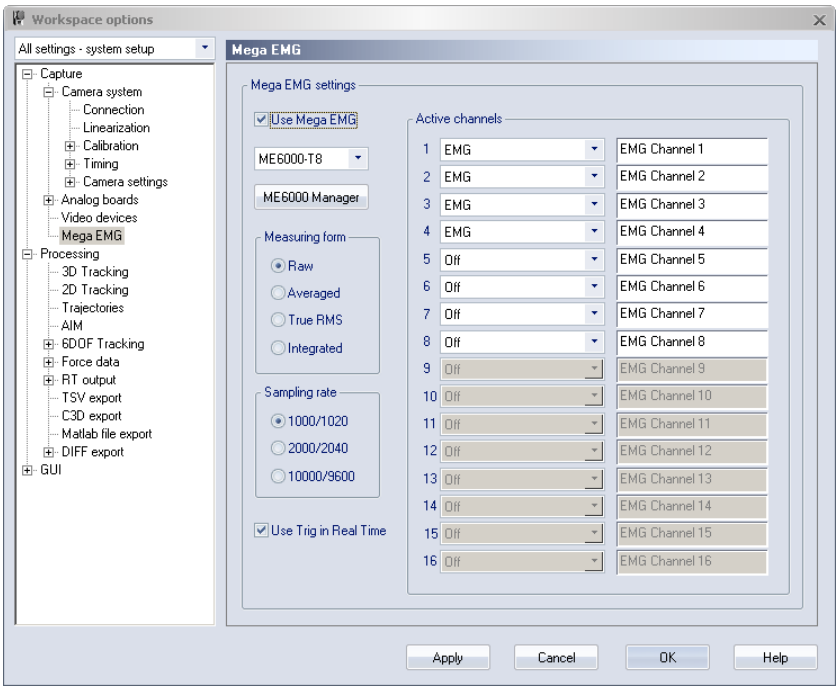
Export Mega EMG data

When the EMG data is exported to C3D format 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.

For MAT and TSV export the EMG data and accelerometer data is exported separately with the original capture frequency.

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 you must enable it on the **Input Devices** page. 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.

Delsys Trigno EMG

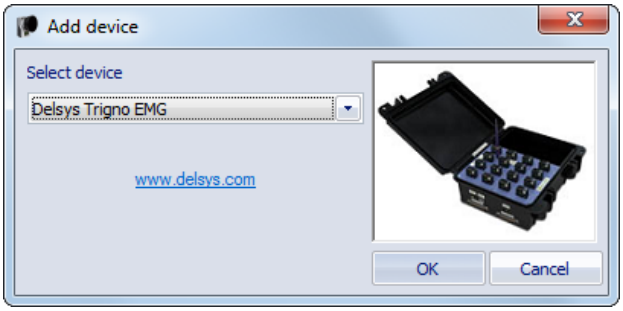
QTM supports integration with Delsys Trigno and Trigno Avanti EMG systems. QTM supports all available types of Trigno and Trigno Avanti sensors.

The following instructions for Delsys Trigno EMG will only cover how to connect to and capture data in QTM. For more detailed information on EMG and Trigno please refer to the Delsys documentation.

Trigno installation

Follow these instructions to install a Delsys Trigno system on a new computer.

1. Connect and install the Delsys Trigno system according to the chapters "Trigno computer installation" below and "Trigno trigger connection" on page D - 28.
2. Start the Trigno system.
3. Go to the **Input Devices** page in the **Project options** dialog.
   a. Click on the **Add Device** option.



   b. Select **Delsys Trigno EMG** from the drop-down list and click OK.
4. Activate the **Delsys Trigno** in the list.
5. Double-click on the **Delsys Trigno** line and go to the **Delsys Trigno** page.
   a. Make sure that the **Connection settings** are working, see chapter "Delsys Trigno QTM settings" on page D - 32
6. Activate the EMG and acceleration channels that you want to use.

Trigno computer installation

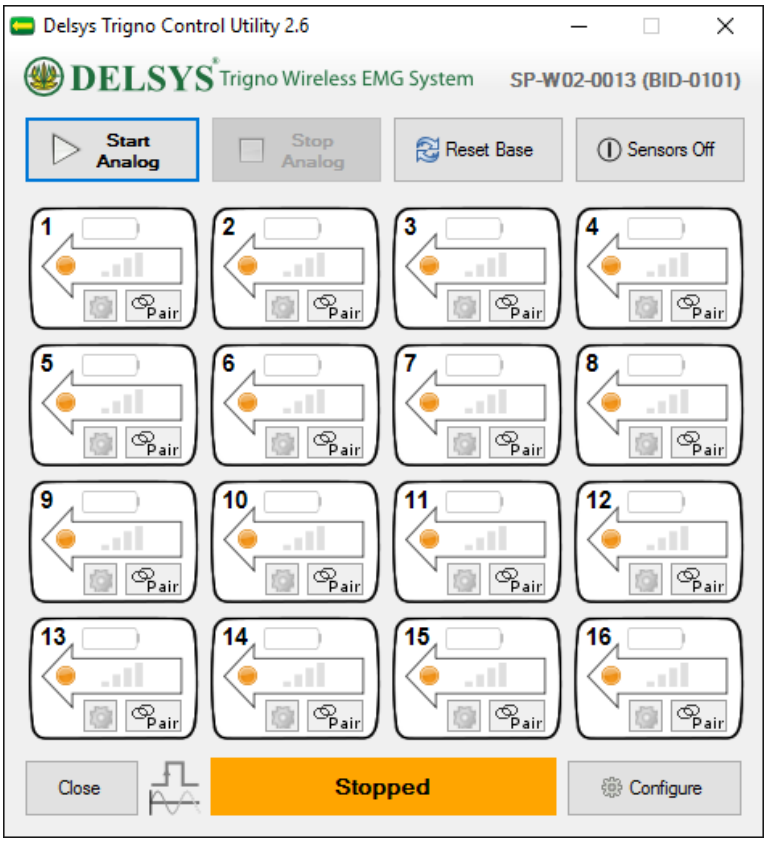
Before you connect the Trigno system to the computer you need to install the Delsys **Trigno Control Utility** program. This is included in the **Trigno SDK**. It is highly recommend to download the SDK via QTM via **Project Options**>**Input Devices**>**Download device drivers** to make sure that you use a compatible version.

Note: Make sure that your Trigno base station and sensors have the firmware installed that is compatible with the Delsys SDK version used for QTM. For upgrading or downgrading firmware, please refer to Delsys documentation or support.

Follow these instructions to start the Delsys Trigno EMG.

1. Connect the power and USB connector of the Trigno Module.
2. Start the **Trigno Control Utility** program.

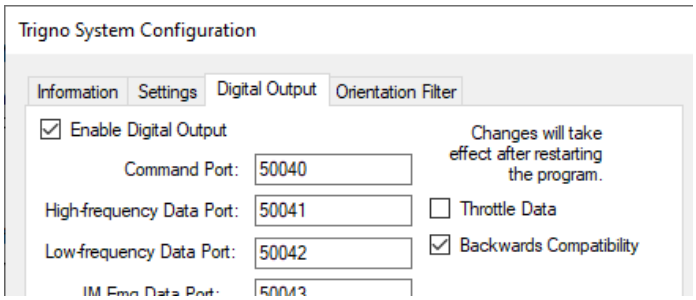
   Note: We recommend to use the **Trigno Control Utility** that comes with the Trigno SDK, even if you have Delsys **EMGworks** software installed. Make sure that you run SensorBaseControl.exe, which is located under C:\Program Files (x86)\Delsys, Inc\Trigno Server 3.5 (or similar).



   a. The first time you use the Trigno system on a computer you need to pair the sensors. Click **Pair** on the sensor images in the program and then hold down the button on sensor for about 3 seconds until it is detected. For Trigno Avanti sensors, the sensor is paired by tapping it over the built-in magnet of the base station, indicated by the lock decal.

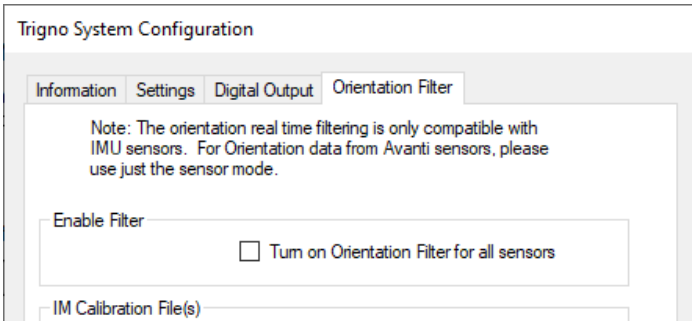
   b. Press the Configure button and make sure that the following settings are correct:

      i. On the tab **Digital Output**, make sure that the option **Backwards Compatibility** is checked.



Warning: Unchecking the **Backwards Compatibility** option will lead to unsynchronized EMG data.

ii. On the tab Orientation Filter, make sure that the option **Turn on Orientation Filter for all sensors** is unchecked.



c. You can configure the sensors (e.g., the gain of the accelerometers) by pressing the configure button of the sensors. For Avanti IMU sensors, make sure to use the **EMG+IMU** mode.

Note: QTM automatically sets the Avanti sensor mode to **EMG+IMU** if the chosen mode is not supported.

d. For more information on how to set up the Trigno system, please refer to the Delsys documentation.

Trigno synchronization connections

For synchronization of the Trigno system with Qualisys the use of the **Delsys Trigger Module** is required. There are two synchronization options:

Use of the **trigger button** for synchronization, see chapter “Trigno trigger connection” below For this option, the use of a trigger button is required.

Use of **Measurement Time** for synchronization, see chapter “Trigno Measurement Time connection” on page D - 30 Use this option if you want to start a measurement without a trigger button.

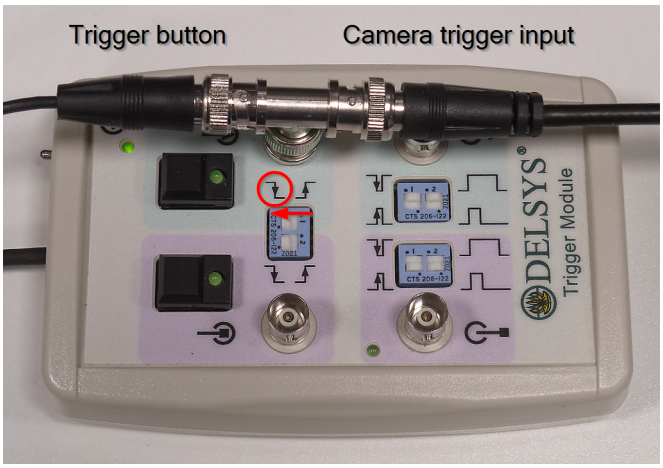
For correct synchronization of the EMG data, it is very important to select the correct **Input** option corresponding to the used hardware connection in the Trigno QTM settings, see chapter “Delsys Trigno QTM settings” on page D - 32.

IMPORTANT: Using the wrong **Input** option in the QTM settings will lead to synchronization errors.

Trigno trigger connection

Follow these instructions to connect the trigger to the Trigno EMG.

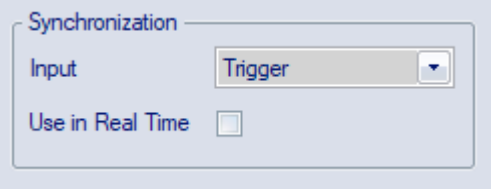
1. Connect the **Delsys Trigger Module** to the **Trigger** port on the **Trigno base station**.



2. Put a BNC T-coupling on the **Start Input BNC** connector (green side of the module) so that you can also connect a trigger button to the input. The measurement must be started with the trigger button.
3. Connect the **Start Input BNC** connector to the trigger input of the camera system.
   a. When using the Camera Sync Unit, use the **Trig NO** input on the Camera Sync Unit.
      Note: The **Trig NC** input on the Camera Sync Unit cannot be used with the Qualisys trigger button.
   b. When using an Oqus camera for synchronization, use the **Trig in** connector on the control port splitter cable.
4. Make sure that you select the correct starting edge with the **Start Input Edge** selector on the module. The default setting is falling edge for the trigger signal on an Oqus system.
   Note: The green LED on the trigger button of the Trigger module lights up when the trigger signal arrives.

In QTM, make sure the synchronization settings are correct.

1. Under **Project Options** go to **Input Devices** > **EMGs**> **Delsys Trigno**. Set **Input** to *Trigger*.

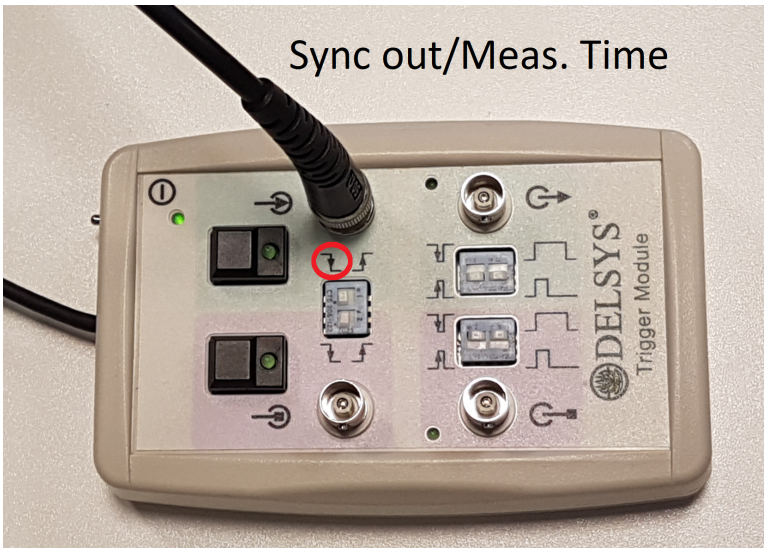


2. Open the **Synchronization** page in the **Project Options** and manage the **Trigger port(s)** settings, see chapter “Trigger ports” on page 156.
   a. For **Oqus**, set **Function** to *Start capture* and make sure that **TTL signal edge** is set to *Negative*.
   b. When using a **Camera Sync Unit**, set **Trig NO: Function** to *Start capture* and make sure that **Trig NO: TTL signal edge** is set to *Negative*.

Trigno Measurement Time connection

Follow these instructions to set up the hardware for synchronization of the Trigno system using **Measurement Time**.

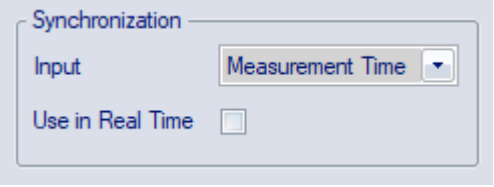
1. Connect the **Delsys Trigger Module** to the **Trigger** port on the **Trigno base station**.



2. Connect the **Start Input BNC** connector (green side of the **Delsys Trigger Module**) to the synchronization output of the Qualisys system.
   a. When using a Camera Sync Unit, use the **Meas. Time** input on the **Camera Sync Unit**
   b. When using an Oqus camera for synchronization, use the **Sync out** connector on the control port **splitter cable**.
3. Make sure that you select the correct starting edge with the **Start Input Edge** selector on the **Delsys Trigger Module**. The default setting in QTM is negative polarity, or falling edge.
   Note: The green LED on the trigger button of the **Delsys Trigger Module** lights up when the synchronization start signal arrives.

In QTM, make sure the synchronization settings are correct.

1. Under **Project Options** go to **Input Devices** > **EMGs**> **Delsys Trigno**. Set **Input** to *Measurement Time*.



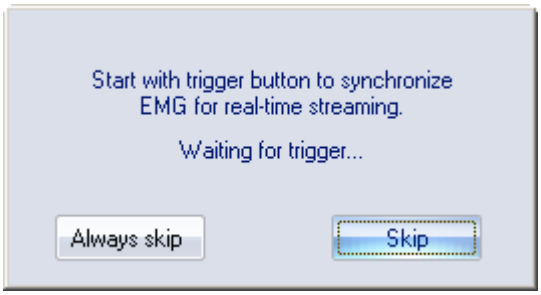
2. Open the **Synchronization** page under **Project Options**.
   a. When using a Camera Sync Unit, make sure that under **Measurement time** the **TTL signal polarity** is set to *Negative*, see chapter "Measurement time (Camera Sync Unit)" on page 165.
   b. When using an Oqus camera for synchronization, select the camera to which the splitter cable is connected. Under **Synchronization Output** set

Mode to *Measurement time*, and make sure that **TTL signal polarity** is set to *Negative*, see chapter “Synchronization output” on page 162.

Making a measurement with Trigno

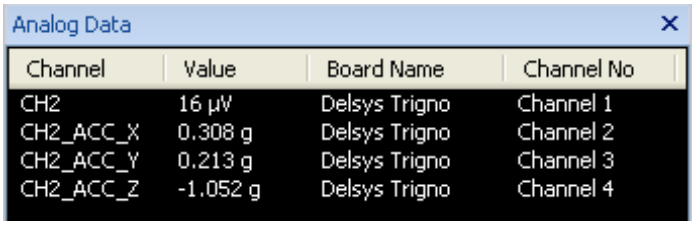
With the Trigno EMG it is possible to capture up to 16 EMG channels including auxiliary data. Follow these steps to make a measurement with Trigno EMG in QTM.

1. The first time a Trigno EMG is connected to a computer you must follow the instructions in the chapter “Trigno computer installation” on page D - 26.
2. Make sure to set up the synchronization correctly according to the instructions in chapter “Trigno synchronization connections” on page D - 28.
3. Make sure that the **Trigno Control Utility** program is running on the computer. This program is needed for the communication with the EMG system.
   a. Check that the EMG sensors that you want to use are active in the program and that they are paired with the correct number.
   b. The EMG+IMU mode is the only supported mode for Avanti sensors. If another mode is selected in **Trigno Control Utility** then the mode is automatically changed to EMG+IMU.
4. Make sure that **Delsys Trigno** is activated on the **Input Devices** page.
5. Go to the **Delsys Trigno** page and select the settings below, for more information see chapter “Delsys Trigno QTM settings” on the next page.
   a. Make sure that QTM has connection with the **Trigno Control Utility** program. If it does there is an OK after the port numbers.
   b. Activate the EMG and accelerometer sensors you want to use in the list. Do not activate sensors that are not needed since it will only result in larger files.
6. Close the **Project options** dialog and start a new measurement.
   a. When **Use Trigger in Real Time** is selected, the Trigno 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 Trigger in Real Time** option on the **Delsys EMG** page. This option requires that the trigger start option is used for synchronization.



7. Open the **Data info window** on the **View** menu and then right-click and select **Display Analog data**.
8. The data from the Delsys EMG is listed with the analog data. The channels are called the **Channel name** for the EMG data. The auxiliary channels are

called the **Channel name** and then _ACC_X and so on. The **Board name** is Delsys Trigno.



Note: The Trigno EMG and auxiliary data is automatically upsampled to 2000 Hz. The original sample frequencies are 1925.925 Hz for EMG data and 148.148 Hz for auxiliary data.

9. When starting a capture the start of the EMG data will be synchronized with the motion data according to the selected synchronization method.

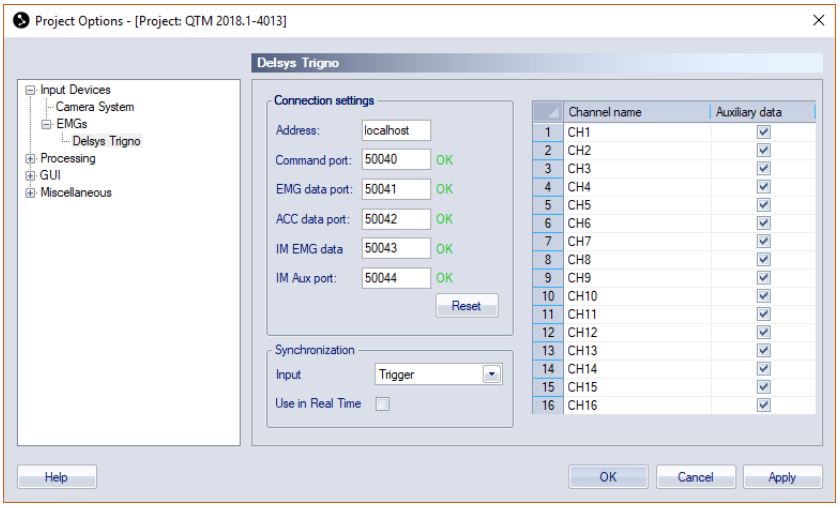
Export Trigno EMG and auxiliary data

The EMG data is resampled at 2000 Hz via the Delsys SDK when recorded in QTM. Auxiliary sensor data is upsampled in QTM to 2000 Hz. For the export to MAT and TSV the sample frequency will be 2000 Hz for all Trigno sensor data.

When you are using C3D export it is recommended to choose a capture frequency with a direct integer relationship to 2000 Hz to avoid additional resampling when exporting to C3D. See chapter “C3D file format” on page 398 for more information.

Delsys Trigno QTM settings

The **Delsys Trigno** page, with settings for the Delsys Trigno EMG, is included in the **Project options** dialog when Delsys Trigno is activated on the **Input Devices** page. To use Delsys Trigno you need to install the drivers that can be found on the **Download device drivers** link on the **Input Devices** page. For more information about how to use Trigno, see chapter “Delsys Trigno EMG” on page D - 26.



The page contains the different options for the Trigno EMG. To change any Trigno settings that are not included on the page, go to the **Trigno Control Utility**.

The following settings are available for Delsys Trigno:

Connection settings

Address
The IP address of the computer that is running the Trigno Control Utility. The default is **localhost** which is the same as the QTM computer.

Command port, **EMG data port**, **ACC data port**, **IM EMG data** and **IM Aux port**
Ports used to communicate with the Trigno Control Utility. The defaults are 50040, 50041, 50042, 50043 and 50044.

Synchronization settings

Input
Select the synchronization input. The selected input depends on which signal from the Qualisys system is used as synchronization signal. The options are:

Trigger: This option should be selected when the trigger button is connected to the Delsys Trigger Module as described in chapter "Trigno trigger connection" on page D - 28. When this option is selected, the trigger delay from the Qualisys system is compensated for.

Measurement Time: This option should be selected when the synchronization or measurement time output of the Qualisys system is connected to the Delsys Trigger Module as described in chapter "Trigno Measurement Time connection" on page D - 30

Use trigger in real time
The **Use trigger 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 trigger in real time** option when the setup is finished to minimize the number of times you have to press the trigger button.

Channels

Channel name
Activate the EMG and Acceleration sensors and enter a name for the channels. Make sure that the channel number matches those that are active in the Trigno Control Utility. The name of the EMG channel is the same as the **Channel name**. The auxiliary data channels have the channel name with a suffix, for example _ACC_X, _ACC_Y and _ACC_Z for the acceleration data.

Auxiliary data
Check this option for the channels for which you want to retrieve auxiliary data from the sensors (e.g., acceleration, gyro, magnetometer). If you do not need the auxiliary data, uncheck it to reduce the amount of analog data in the QTM files.

Cometa EMG

QTM supports integration with the Cometa Wave Plus wireless EMG system. The supported sensor types are EMG and IMU sensors. A single Cometa base unit allows the use of maximum 16 EMG and/or IMU sensors. It is possible to connect two base units for the use of maximum 32 sensors.

Note: Refer to Cometa documentation or support for how to set up two base units.

For use with QTM the Cometa system should include the following components:

1. A base unit
2. A docking base with power supply
3. EMG and/or IMU sensors
4. A trigger box with BNC connector

The following instructions will cover how to connect to and capture Cometa EMG and IMU data in QTM. For more detailed information please refer to the Cometa documentation.

There is also a tutorial available at QAcademy for connecting and using Cometa EMG with QTM.

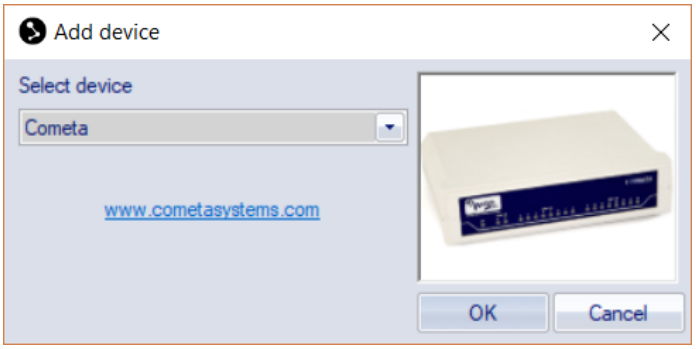
Cometa installation

Before using the Cometa EMG system for the first time you will need to install the drivers. You can download the current drivers via QTM via **Project Options**>**Input Devices**>**Download device drivers**. Follow these steps to install the drivers:

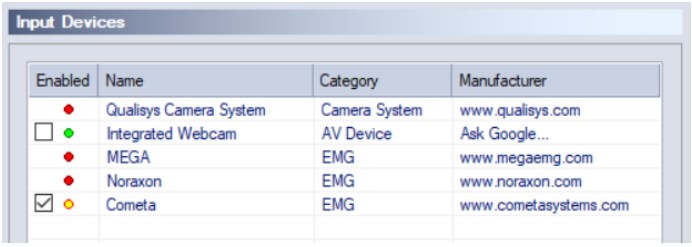
1. Connect the Cometa base unit to a USB port on the QTM computer. Windows will automatically detect the device.
2. Unzip the file containing the drivers (typically called **emgmusbdrivers_...zip**).
3. In Windows explorer, locate the folder containing the files that are compatible with your computer (e.g. **Win10\x64**).
4. Right-click on the file **EmgMUsb.inf** and choose **Install** from the context menu.

To add the Cometa device as an input device in QTM, follow these steps:

1. Open the **Input Devices** page in the **Project Options** dialog.
2. Click on the **Add Device** option.
3. Select **Cometa** from the drop-down list and click **OK**.



4. Select **Cometa** in the Input Devices list and click **OK**.



Cometa synchronization setup

For synchronizing Cometa with Qualisys motion capture data, the following accessories are required:

1. A **Cometa trigger box** with BNC connector.
2. A **Camera Sync Unit** (Miqus or mixed Qualisys system) or an **Oqus Sync/Trig splitter cable** (Oqus only system).

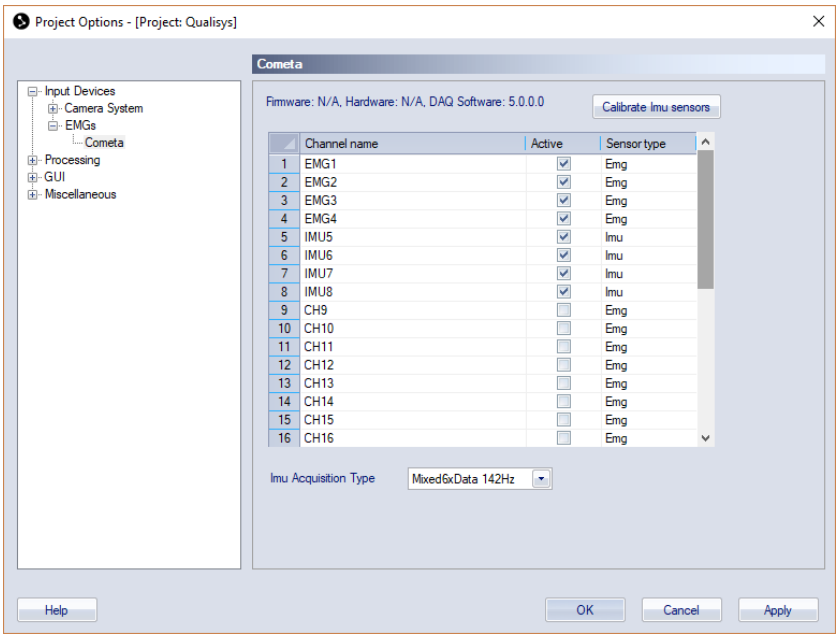
The synchronization is based on the Measurement Time synchronization output of the Qualisys system. There are two options, depending on your Qualisys system.

1. For a Qualisys system including a Camera Sync Unit:
   a. Connect the **MEAS. TIME** output of the **Camera Sync Unit** with a BNC cable to the **BNC connector** of the **Cometa trigger box**.
2. For an Oqus system without Camera Sync Unit:
   a. Connect the **Sync/Trig splitter cable** to the **control port** of one of the Oqus cameras.
   b. Connect the **Sync out** connector of the **Oqus splitter cable** with a BNC cable to the **BNC connector** of the **Cometa trigger box**.
   c. In QTM open the **Project Options** dialog and go to the **Synchronization** page. Select the camera used for synchronization and set the **Mode** option under **Synchronization output** to **Measurement time**, see chapter "Synchronization output" on page 162.

Synchronization output	
Mode	Measurement time
TTL signal polarity	Negative

Cometa QTM settings

The **Cometa** page, with settings for the Cometa EMG, is included in the **Project options** dialog when Cometa is activated on the **Input Devices** page.



The following settings are available for Cometa EMG.

Calibrate Imu sensors
Press to calibrate the used IMU sensors to compensate for gyroscope and accelerometer offset. When using fused data, the current orientation of the sensors is set as start position.

Channels

Channel name: Name of the channel. Click in the text area to edit.

Active: Check sensors that are currently used for data acquisition in QTM.

Sensor type: Select the type of sensor (**Emg** or **Imu**) per channel.

Imu Acquisition Type
Choose the data type for the used IMU sensors. The options are:

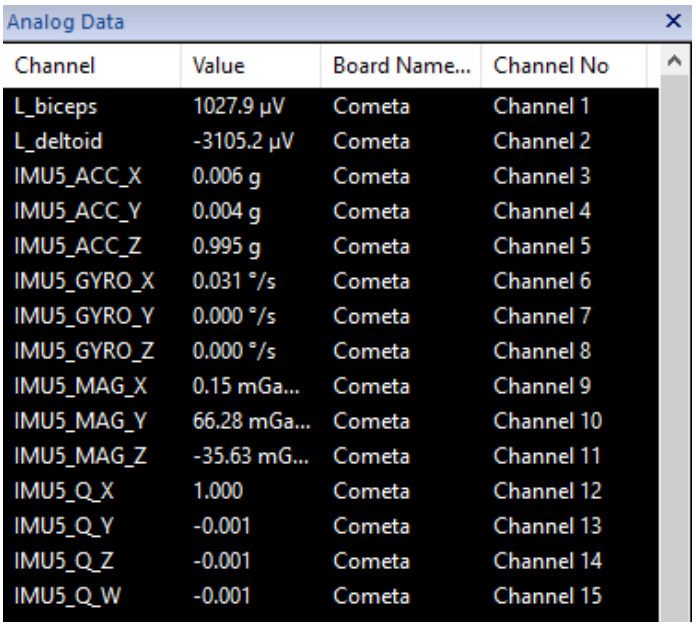
RawData 284Hz: Acquisition of 3-axis accelerometer (**ACC_X**, **ACC_Y**, **ACC_Z**), gyroscope (**GYRO_X**, **GYRO_Y**, **GYRO_Z**) and magnetometer (**MAG_X**, **MAG_Y**, **MAG_Z**) data at a sample rate of approximately 284 Hz (sample ratio 1/7 of 2000 Hz).

Mixed6xData 142Hz: Acquisition of raw and fused data at a sample rate of approximately 142 Hz (sample ratio 1/14 of 2000 Hz). In addition to the raw data, quaternions (**Q_X**, **Q_Y**, **Q_Z**, **Q_W**) are included representing the orientation of the IMU sensors.

Making a measurement with Cometa

For making a measurement with Cometa in QTM, follow these steps.

1. Set up the Cometa system according to the instructions in chapter “Cometa installation” on page D - 34.
2. Set up the synchronization according to the instructions in chapter “Cometa synchronization setup” on the previous page.
3. Take the sensors to be used from the docking base and check that the LEDs of the assigned channels on the base unit are lighting up.
4. Select the used channels in the QTM settings and make sure that the sensor type is set correctly, see chapter “Cometa QTM settings” on the previous page.
5. When using IMU sensors, make sure to select the correct IMU acquisition type and calibrate the sensors before starting a new series of measurements. It is recommended to regularly recalibrate the sensors. Refer to Cometa documentation for detailed recommendations.
6. Press **New** or **Ctrl+N** to start a measurement.
7. For viewing the EMG and IMU data, open a **Data Info window** in the **View** menu, right-click and select **Display Analog data**.
8. The data from the Cometa EMG is listed with the analog data. The channels are called the **Channel name** for the EMG data. The IMU channels are called the **Channel name** and then _ACC_X and so on. The **Board name** is Cometa.



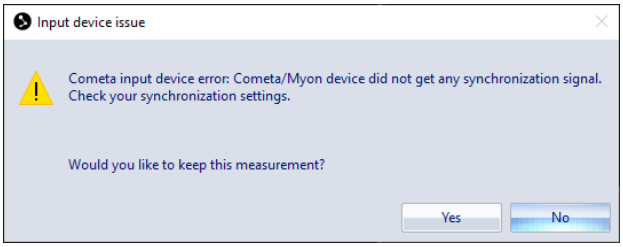
Analog Data

Channel	Value	Board Name...	Channel No
L_biceps	1027.9 µV	Cometa	Channel 1
L_deltoid	-3105.2 µV	Cometa	Channel 2
IMU5_ACC_X	0.006 g	Cometa	Channel 3
IMU5_ACC_Y	0.004 g	Cometa	Channel 4
IMU5_ACC_Z	0.995 g	Cometa	Channel 5
IMU5_GYRO_X	0.031 °/s	Cometa	Channel 6
IMU5_GYRO_Y	0.000 °/s	Cometa	Channel 7
IMU5_GYRO_Z	0.000 °/s	Cometa	Channel 8
IMU5_MAG_X	0.15 mGa...	Cometa	Channel 9
IMU5_MAG_Y	66.28 mGa...	Cometa	Channel 10
IMU5_MAG_Z	-35.63 mG...	Cometa	Channel 11
IMU5_Q_X	1.000	Cometa	Channel 12
IMU5_Q_Y	-0.001	Cometa	Channel 13
IMU5_Q_Z	-0.001	Cometa	Channel 14
IMU5_Q_W	-0.001	Cometa	Channel 15

9. Start a capture via the **Capture dialog** (**Ctrl+M**). The start of the EMG and IMU data will be synchronized with the motion data.

   Note: In Preview mode there will be a certain latency of the EMG and IMU data.

   Note: When using hardware synchronization and the synchronization signal is not received by the Cometa device, QTM will show the following warning after the capture has been finished. When saving the measurement the data will not be synchronized.



10. EMG and IMU data can be exported to several export formats, see chapter “Export Cometa EMG and IMU data” below.

Export Cometa EMG and IMU data

Cometa EMG and IMU data can be exported to TSV, MAT and C3D file formats. For the TSV and MAT export, make sure that the **Analog** data type is selected.

The sample frequency of all Cometa EMG and IMU data is 2000 Hz. The actual sample rate of IMU data depends on the used IMU acquisition type, see chapter “Cometa QTM settings” on page D - 35.

When you are using C3D export it is recommended to choose a capture frequency with a direct integer relationship to 2000 Hz to avoid additional resampling when exporting to C3D. See chapter “C3D file format” on page 398 for more information.

Appendix E: Eye tracking hardware

This appendix contains information about the Eye tracking hardware integrated in QTM, see “Eye tracking hardware in QTM” below.

Eye tracking hardware in QTM

There are two levels of integration for the eye trackers. Some eye trackers are integrated in QTM so their data is captured with the rest of the data. Then there are other eye trackers where the data needs to be captured in their own program and then the data can be combined in other software such as Matlab.

Integrated eye trackers in QTM

Note: QTM requires the **EyeTracker** plugin to capture from the Dikablis and SMI eye tracker. Contact Qualisys AB if you want to buy the plugin.

Dikablis - www.ergoneers.com
The eye tracker data from Dikablis can be captured directly in QTM. The gaze vectors can then be viewed and exported. The settings and how to handle the data is described in the chapter “Dikablis eye tracker” on the next page.

SMI - www.smivision.com
The gaze vector data from SMI can be captured directly in QTM. The gaze vectors can then be viewed and exported. The settings and how to handle the data is described in the chapter “SMI Eye Tracker Glasses” on page E - 15.

Tobii - www.tobii.com
The gaze vector data from SMI can be captured directly in QTM. The gaze vectors can then be viewed and exported. The settings and how to handle the data is described in the chapter “Tobii eye tracker” on page E - 24.

Eye trackers captured in external software

ASL - www.asleyetracking.com
ASL has made a integration called EyeHead to their Mobile Eye EyeVison program that retrieves 6DOF data from QTM in real time. In addition to head tracking, the ASL integration can also calculate gaze vectors and intersection points for gaze vectors and surfaces. For a basic description of the integration see chapter “ASL EyeHead integration” on page E - 34. For any detailed question about the ASL integration please refer to ASL support or their manual.

Smart-Eye - smarteye.se
For the Smart-eye hardware it is possible for the Qualisys system to lock on

the capture frequency of the Smart Eye camera. Then the start synchronization is handled via a real time event from QTM that is sent to the Smart eye software. Finally the Smart Eye and Qualisys systems can be aligned via markers on the Smart eye calibration equipment. For questions about the Smart eye integration please contact support@qualisys.com.

Dikablis eye tracker

QTM has built-in support for the Dikablis Essential and the Dikablis Professional eye trackers from Ergoneers, allowing to view and measure 3D gaze vector data in QTM.

To use Dikablis with QTM it is required that you run the Dikablis D-Lab software on another computer. The following chapters describe how to use Dikablis in QTM. For any questions about how to use the Dikablis software, please contact the Dikablis support.

Dikablis installation

Eye tracking with Dikablis in QTM requires the use of Dikablis software on a separate computer. QTM supports Dikablis **D-Lab** (version 3.50 or later). Please contact Dikablis if you need help with the installation on a computer.

The only connection that is needed for Dikablis is that the computers must be on the same network. Check with your IT administrator if you don't know how to achieve this.

Add Dikablis eye tracker to a project

The Dikablis eye tracker can be captured together with all the other data in QTM. Follow these instructions to add Dikablis to your project:

1. Make sure that the Dikablis computer is connected on the same network as the QTM computer.

   Note: The Dikablis eye tracker has no synchronization options so you do not need to connect any other signals. This means that data from Dikablis is only synchronized via a start command, but you can modify the delay between Dikablis and the marker data in QTM to find the best option.

2. Start the Dikablis software.

   a. For the D-Lab software, follow the instructions in “Setting up eye tracking in D-lab” on page E - 4.

   b. For more information on the use of the Dikablis eye trackers and software, please refer to the Dikablis manual or contact Ergoneers support.

3. You need to make sure that you have a rigid body on the Dikablis eye tracker and that it is defined in QTM. If you haven't done this before follow these steps:

   a. Put at least 4 markers on the eye tracker, it is good if the markers are on both sides of the eye tracker. For more information on how to place markers for a rigid body, see chapter “How to design a 6DOF body” on page 337.

   b. Place the eye tracker with the rigid body in the measurement volume. It is recommended that you have it on a person for the best definition.

c. Make a short measurement, about 3 seconds, with the person standing still looking straight forward.

d. Select the markers of the rigid body. Define the body by right-clicking on them and selecting **Define rigid body (6DOF)**. You can give it any name you want.

e. Start preview with **New** on the **File** menu.

f. Open **Project options** and go to the **Processing** page. Turn on the **Calculate 6DOF** and **Calculate gaze vector data** option for both **Real time actions** and **Capture actions**.

g. You may want to rotate the coordinate system with the **Rotate** option for visual reasons, but this does not affect the quality of the gaze vector data. If so go to the **6DOF tracking** page.

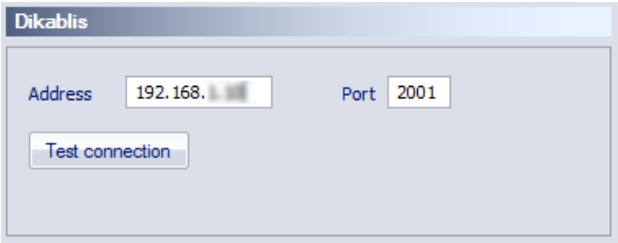
⚠ **IMPORTANT**: The rigid body definition must not be changed after you calibrate the gaze vector. That will make the gaze vector data useless.

4. To add the Dikablis eye tracker device, open the **Input Devices** page in the **Project options** dialog and click on **Add device**.

a. Select the **Dikablis Eye Tracker** from the list of devices and click OK.
Note: QTM requires the **EyeTracker** plugin to capture data with the Dikablis eye tracker. Contact Qualisys AB if you are interested in purchasing the plugin.

5. Make sure that the Dikablis device is enabled in the list. Then double-click on the **Dikablis** line in the Input devices list to open the **Dikablis** page.

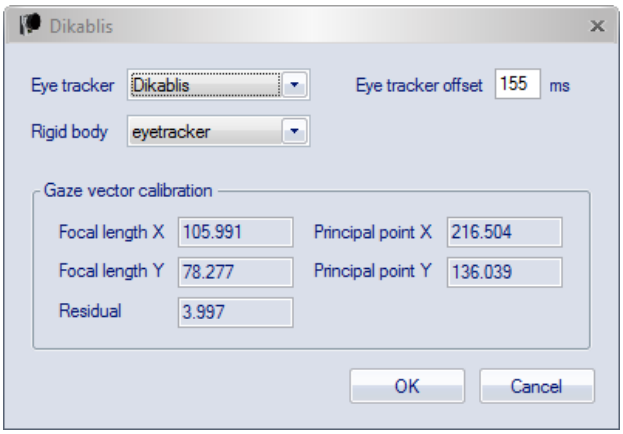
6. On the **Dikablis** page enter the following information.



Address
The IP address or name of the Dikablis computer.

Port
The port used in the Dikablis program. The default port in D-Lab Mocap Tool is 2001.

7. Then use **Test connection** to make sure that QTM can communicate with the D-Lab Mocap Tool.
8. Go to the **Gaze Vector** page to add a gaze vector.
   a. Click on **Add** to add a new gaze vector.



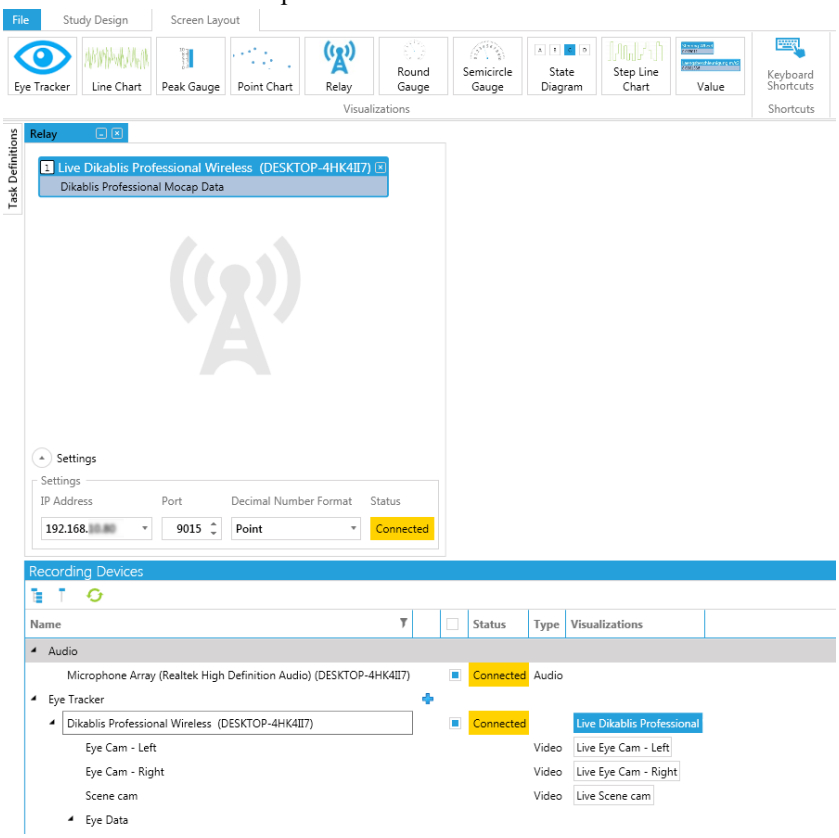
   b. Select **Dikablis** on the **Eye Tracker** option.
   c. Select the rigid body that you created for the eye tracker on the **Rigid body** option.
9. The final step is to calibrate the gaze vector, see chapter “Dikablis gaze vector calibration” on page E - 6.

Setting up eye tracking in D-lab

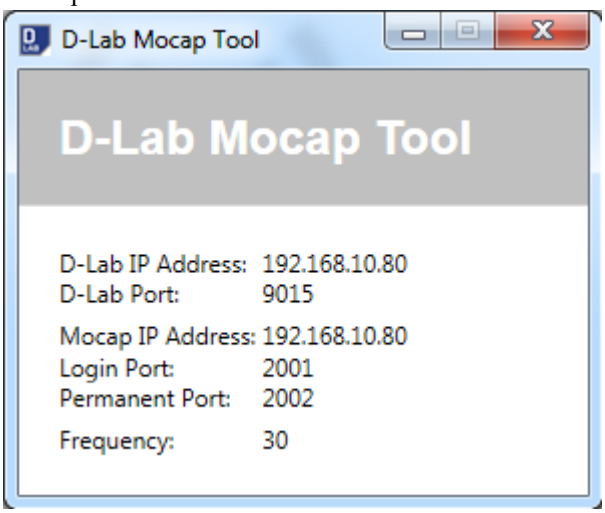
To set up the eye tracker in Dikablis D-lab 3.0 software, follow these steps. Please also refer to the documentation from Ergoneers, which can be downloaded here.

1. Start D-Lab.
2. Create a new study or open an existing one.
3. Change to screen layout. The eye tracking device should be visible under Recording Devices as "connected".
4. Add a Relay window by clicking on the Relay button. Select the correct IP address. The Dikablis and the QTM computer should be connected to the

same local network. The port number can be left to default 9015.

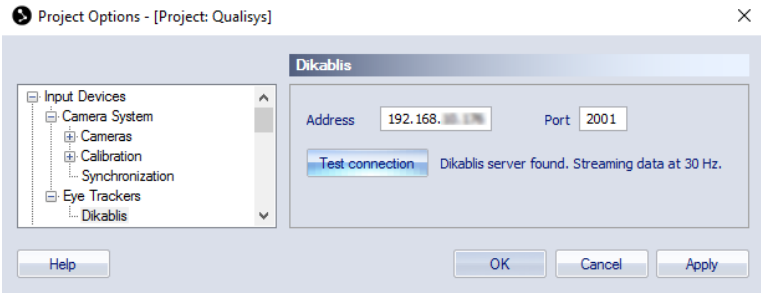


5. Drag and drop the eye tracker from Recording Devices. It is named Dikablis Professional Wireless in the picture above.
6. Setup the **D-Lab Mocap Tool** by editing the **D-Lab.Mocap.exe.config** file located in the D-Lab install path. Set **DLabIP** to the ip address of your D-Lab computer. Set **MocapIP** to the ip address of the computer running the D-Lab Mocap Tool. Usually the same as the D-Lab computer. Set **Frequency** to Your Dikablis eye tracker capture rate.
7. Start D-Lab.Mocap.exe as Administrator and leave it running. It will act as a proxy between D-Lab and QTM. D-Lab.Mocap.exe is located in the D-Lab install path.



Optionally, you can check if the streaming is set up correctly.

1. Monitor the streaming of the values in D-Lab by adding a Chart or a Value window and dragging Pupil X and/or Pupil Y of the left eye into it.
2. Test the connection in Dikablis settings in QTM (see “Dikablis QTM settings” on page E - 13).

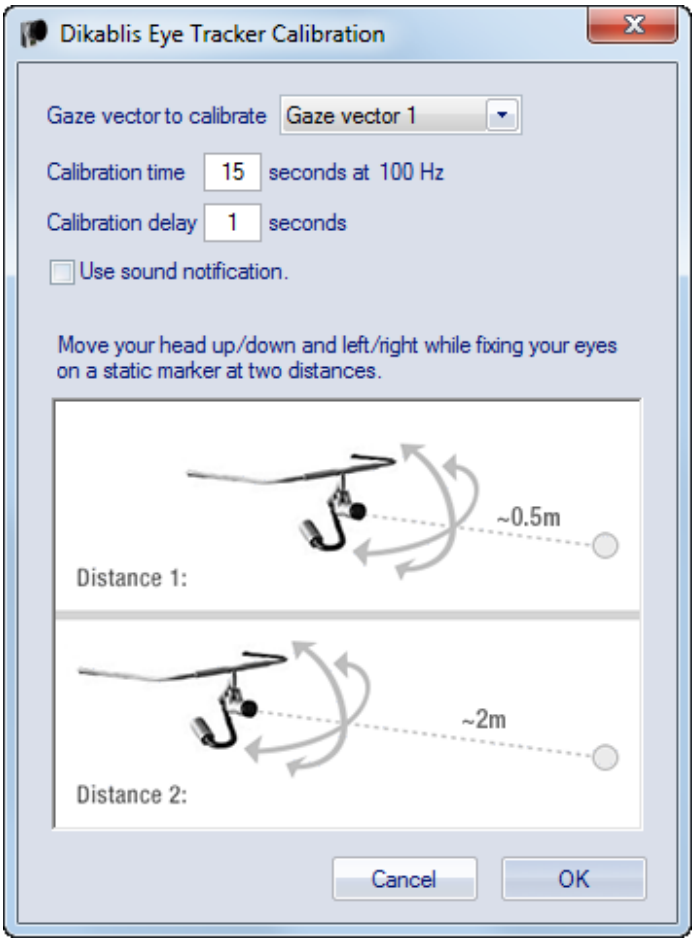


Dikablis gaze vector calibration

The Dikablis eye tracker data must be related to the rigid body to get a gaze vector. This is done by making a gaze vector calibration that finds the best fit between the eye tracker data and the rigid body. This process must be performed as soon as the eye tracker has moved in relation to the head. Therefore it is recommended that it is performed at least both before and after a measurement session. If you suspect the eye tracker has moved during a measurement session it is best to do calibration right away.

Follow these steps to perform a calibration:

1. First of all the Dikablis device and the Gaze vector must exist in the project, see chapter “Making a measurement with Dikablis” on page E - 8.
2. Start preview in QTM with **New** on the **File** menu.
3. Place a single marker somewhere in the measurement volume.
   a. Place the marker in level with the eyes. This will make it easier to move the head to achieve a good calibration.
   b. Make sure that there is just one static trajectory in the 3D view. If there are other static they must be removed so that the gaze vector calibration knows which the reference marker is.
4. Start the gaze vector calibration with **Calibrate/Gaze Vector** on the **Capture** menu.



a. Select the **Gaze vector to calibrate**.

b. Enter the **Calibration time** in seconds that you want to use for the calibration, 15 to 20 seconds is usually enough time to make the calibration.

c. You may use a **Calibration delay** to delay the start of the calibration from when you click OK.

5. When you start the calibration it is important that the subject follow these rules. There are images in the **Dikablis Eye Tracker Calibration** dialog that explains how you are supposed to move the head.

   a. The subject must keep the eyes focused on the single marker during the whole calibration.

   b. The subject must move the head in a circle so that the eyes moves in their socket.

   c. Finally the head movement must be performed on two distances. One that is fairly close about 50 cm and one that is further away around 2 m. If the distances are too far away from these values, then the calibration will fail.

6. If the calibration succeeds the result of the calibration is a residual. This residual should ideally be around 3 to 4 pixels, but it depends among other things on how good that the subject can focus on the marker.

7. The calibration also automatically sets the **Eye tracker offset** setting for the Dikablis eye tracker. The offset is displayed on the **Gaze vector** settings in the **Project options** dialog, see chapter “Dikablis QTM settings” on page E - 13.

The Dikablis eye tracker is now ready for measurements, see chapter “Making a measurement with Dikablis” below.

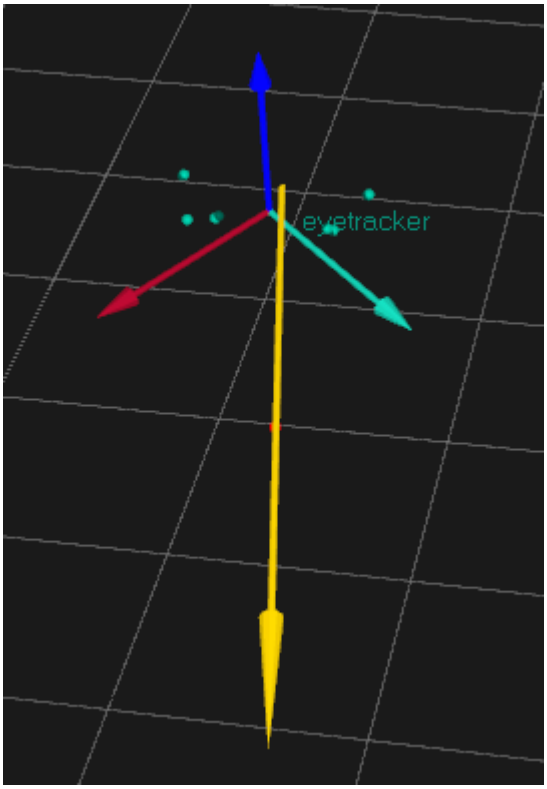
Making a measurement with Dikablis

The eye tracker data is collected together with the marker data in QTM and can be reprocessed and exported together with the other data, see chapter “Process and export Dikablis gaze vector data” on the facing page. Follow these steps to capture eye tracker data.

1. Enable the Dikablis eye tracker on the Input devices page and add a gaze vector on the **Gaze Vector** page, see chapter “Add Dikablis eye tracker to a project” on page E - 2.
2. Calibrate the gaze vector, see chapter “Dikablis gaze vector calibration” on page E - 6

   ⚠ **IMPORTANT:** The eye-tracker must be calibrated as soon as it might have moved. Therefore if there are a lot of head movement in a measurement it is actually recommended that you calibrate between each measurement. But when there are less head movement it is enough to do a calibration for each measurement session.
3. Start preview in QTM and the gaze vector is displayed in the 3D view window as soon as the 6DOF body is visible.

   Note: Make sure that you have activated **Calculate gaze vector data** and **Calculate 6DOF** on the **Processing** page in **Project options**.



4. You can also view the Eye tracker data and Gaze vector data in the **Data info** window, see chapter “Dikablis data in QTM” on page E - 12.
5. The eye-tracking data is captured automatically with the other data in QTM. Since there is no trigger or sync connection on the Dikablis the capture is started via software. This means that the synchronization cannot be known for sure. However an approximate delay for the Dikablis data is calculated automatically in the eye-tracker calibration and that delay is then used for the following captures.
6. Depending on the eye tracking device, the frequency is in the range of 25-60 Hz. This means that the eye-tracker data will mostly update less often than the marker data. So it might look like it is more out sync than it actually is.

Process and export Dikablis gaze vector data

The Gaze vector data can be reprocessed in a file, both in reprocessing and batch processing. This is useful if you need to update the gaze vector data because of changed 6DOF data or a new Gaze vector calibration.

If you have changed the 6DOF data then you need to reprocess the Gaze vector data to update it. Follow these steps to reprocess it.

1. Click on **Reprocess** on the **Capture** menu.
2. Then select **Calculate gaze vector data** in the **Processing steps**.
   a. If you need to change the rigid body that is associated with the gaze vector, then go to the **Gaze vector** page and double click on the gaze vector in the list.
3. Click **OK** in the File reprocessing dialog.

To change the Dikablis calibration then you must use the Batch processing option instead. Follow these steps:

1. Open a file where the gaze vector has the Dikablis calibration that you want to use.
2. Click on **Batch process** on the **File** menu.
3. Select the files that you want to change the calibration in. It is important that the name of the Rigid bodies are the same as the currently open file.
4. Then select **Calculate gaze vector data** in the **Processing steps**.
5. It is also recommend to use the **Auto-backup files before reprocessing**. If you don't make a backup then you cannot revert the data of the file unless you have kept the Dikablis calibration file.
6. Make sure that you change the **Gaze vector settings from** option to **present file**.
7. Click **OK** in the **Batch process** dialog.

The Gaze vector data can then be exported and analyzed in external programs. You can either export it to TSV or MAT files.

TSV

To export the eye tracker data select the **Export eye tracker data if available** option for the TSV export. The result is two TSV files, *_e.tsv and *_g.tsv with eye tracker respectively gaze vector data. The two files includes the following data:

_e.tsv

EYE_TRACKER_NAME
The name of the eye tracker device.

NO_OF_SAMPLES
The number of samples in the eye-tracker data.

FREQUENCY
The frequency of the eye-tracker data.

TIME_STAMP
Date and time when the motion capture file was made. The date and time is followed by a tab character and then the timestamp in seconds from when the computer was started.

WARNING: The time stamps indicate the start of the capture according to the QTM software. The time stamps may not exactly correspond to the first frame of the capture and it is discouraged to use them for synchronization with data from other devices connected to the same computer.

START_OFFSET
The offset time for the first eye-tracker frame. It is needed since you may not always cut the measurement range at the start of eye-tracker frame. Reduce the start time of the eye-tracker data with the offset so that it actually starts before the marker data.

Pupil data
Columns with the pupil data in the following order, X Pos, Y Pos, Width, Height

_g.tsv

GAZE_VECTOR_NAME
The name of the gaze vector.

NO_OF_SAMPLES
The number of samples in the gaze vector data.

FREQUENCY
The frequency of the gaze vector data.

TIME_STAMP
Date and time when the motion capture file was made. The date and time is followed by a tab character and then the timestamp in seconds from when the computer was started.

START_OFFSET
The offset time for the first gaze vector frame. It is needed since you may not always cut the measurement range at the start of gaze vector frame. Reduce the start time of the gaze vector data with the offset so that it actually starts before the marker data.

Gaze vector data
Columns with the gaze vector data in the following order: X Pos, Y Pos, Z Pos, X Vec, Y Vec, Z Vec

MAT
To export the eye tracker data select the **Eye tracker** option for the MATLAB file export. The struct array of the MAT file then includes the following data:

GazeVector

GazeVectorName
The name of the gaze vector.

NrOfSamples
The number of samples in the gaze vector data.

StartOffset
The offset time for the first gaze vector frame. It is needed since you may not always cut the measurement range at the start of gaze vector frame. Reduce the start time of the gaze vector data with the offset so that it actually starts before the marker data.

Frequency
The frequency of the gaze vector data.

GazeVector
Array with the gaze vector data in the following order: X Pos, Y Pos, Z Pos, X Vec, Y Vec, Z Vec

EyeTracker

EyeTrackerName
The name of the eye tracker device.

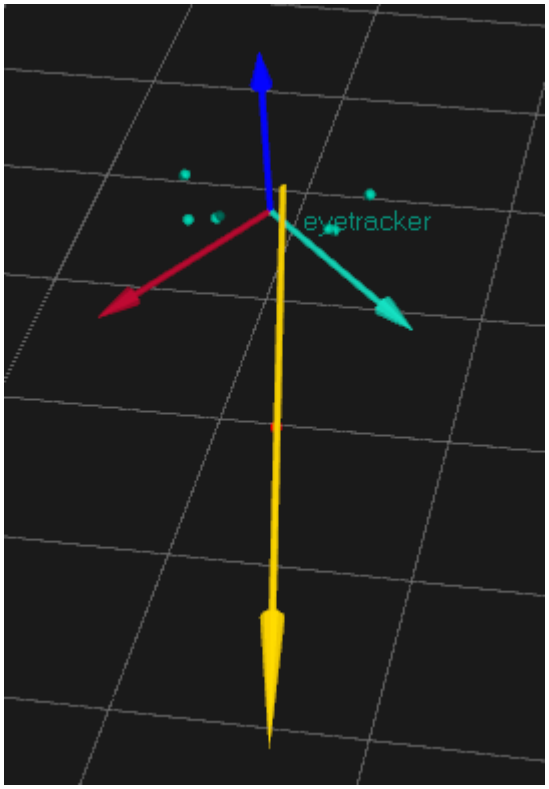
NrOfSamples
The number of samples in the eye-tracker data.

StartOffset
The offset time for the first gaze vector frame. It is needed since you may not always cut the measurement range at the start of gaze vector frame. Reduce the start time of the gaze vector data with the offset so that it actually starts before the marker data.

Frequency
The frequency of the eye-tracker data.

PupilPostion
Array with the pupil data in the following order, X Pos, Y Pos, Width, Height

Dikablis data in QTM



The Gaze vector data is displayed in the 3D view of the window as a vector, for example color and length can be modified on the **3D view settings** page in **Project options**.

There are two types of Dikablis eye tracker data in QTM, these are the eye-tracker data and gaze vector data.

Eye Tracker Data

Device	X pos	Y pos	Width	Height
Dikablis	163	156	34	31

Eye-tracker data
The eye-tracker data is the raw data sent from the Dikablis Recorder program. The data includes:

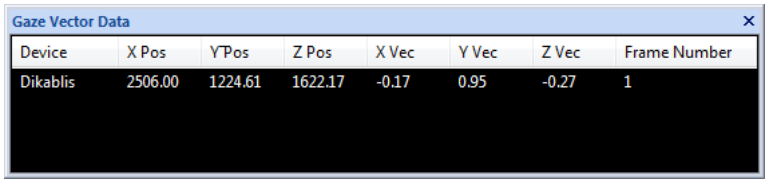
Device
The device is the name of the Eye tracker input device.

X pos
This is X position for the pupil.

Y pos
This is Y position for the pupil.

Width
This is Width for the pupil.

Height
This is Height for the pupil.



Gaze Vector Data

Device	X Pos	Y Pos	Z Pos	X Vec	Y Vec	Z Vec	Frame Number
Dikablis	2506.00	1224.61	1622.17	-0.17	0.95	-0.27	1

Gaze vector data
The gaze vector data is the calculated gaze vector from the eye-tracker and 6DOF data. The data includes:

Device
The device is the name of the Gaze vector.

X Pos
This is the X position for the origin of the Gaze vector in the Global coordinate system.

Y Pos
This is the Y position for the origin of the Gaze vector in the Global coordinate system.

Z Pos
This is the Z position for the origin of the Gaze vector in the Global coordinate system.

X Vec
This is the X value of the Gaze vector in the Global coordinate system.

Y Vec
This is the Y value of the Gaze vector in the Global coordinate system.

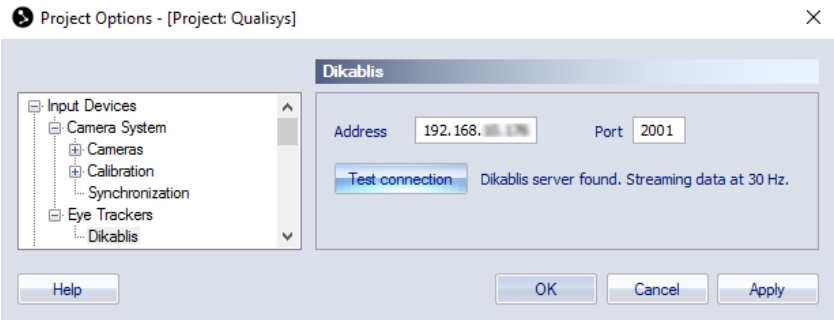
Z Vec
This is the Z value of the Gaze vector in the Global coordinate system.

Frame Number
This is the frame number of the Dikablis data. Since the frame rate of the Dikablis data is lower than the marker data, it means that the Dikablis data is updated less often than the marker data.

Dikablis QTM settings

There are two pages in the **Project options** dialog that have settings related to the Dikablis eye tracker.

The **Dikablis** page contains all of the settings for the communication with the Dikablis software. It is located under **Input Devices/Eye Trackers** in the tree and can also be accessed by double clicking on the Dikablis entry in the Input Devices list.



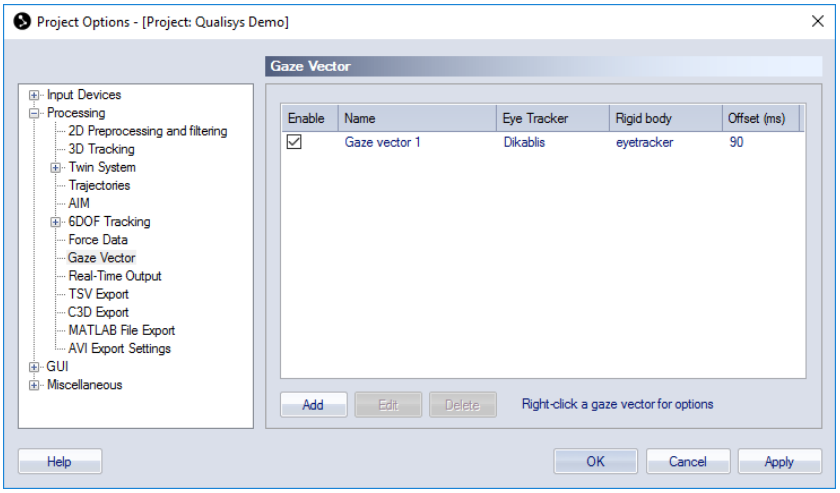
The **Dikablis** page contains the following settings.

Address
Ip address of the computer running the Dikablis software.

Port
The port used for communication with the Dikablis software (D_Lab Mocap Tool). The default port is 2001.

Test connection
Use this option to test that there is communication with the Dikablis software. It must be up and running for the test to work.

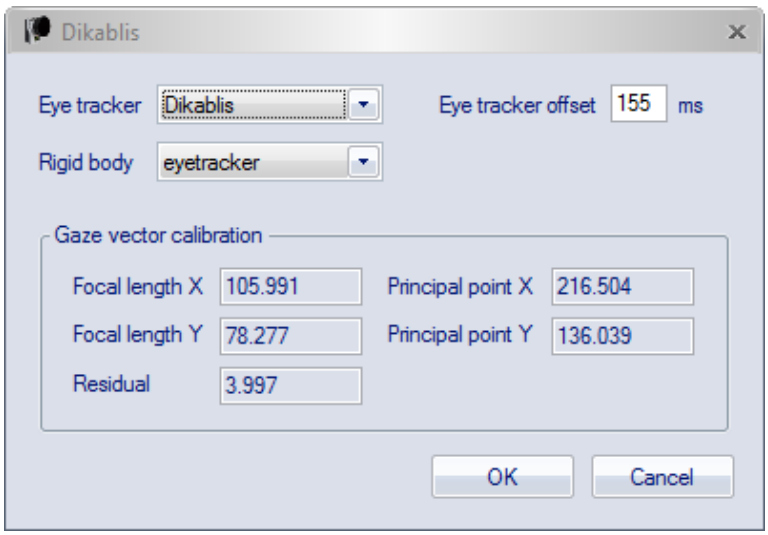
The second page with Dikablis related settings is the **Gaze Vector** page. It is located under **Processing** in the tree.



The **Gaze Vector** page contains a list of the current gaze vectors and the settings needed to calculate a gaze vector from the eye tracker data. Use the following options to modify the list.

Add
Add a new gaze vector to the list.

Edit (or double-click on a gaze vector)
These two alternatives open the dialog below for the selected gaze vector.



The options in the dialog defines how to calculate the Gaze vector.

Eye tracker
Select an eye tracker from the list.

In **Project options** the list contains all of the eye trackers that are available on the **Input devices** page. For reprocessing with file settings the list contains all of the eye trackers that were used in the capture.

Eye tracker offset
The **Eye tracker offset** is the number of milliseconds that the eye tracker data is delayed compared with the marker data. This value is then used to compensate to align the data in the file.

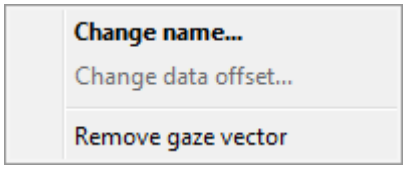
Rigid body
Select the rigid body that is mounted on the eye tracker. This is used as reference for the gaze vector calibration.

Note: In **Project options** the list contains all of the rigid bodies that are available on the **Input devices** page. For reprocessing with file settings the list contains all of the eye trackers that were used in the capture.

Gaze vector calibration
This sector displays the current calibration values for the gaze vector. If there is no calibration then the values are empty. For more information on how to calibrate and reprocess the calibration see chapters "Dikablis gaze vector calibration" on page E - 6 and "Process and export Dikablis gaze vector data" on page E - 9.

Delete
Delete the selected gaze vector.

Right-click to open menu



From the menu you can **Change name** of the selected gaze vector and **Remove gaze vector**.

SMI Eye Tracker Glasses

The SMI Eye Tracker Glasses is integrated with QTM so that you can view and export gaze vector data. To use SMI it is required that you have the SMI iView ETG program running on another computer. The following chapters will describe how to use SMI in QTM. For any questions about how to use the SMI program please contact the SMI support.

SMI eye tracker installation

The SMI iView ETG program is installed on a separate computer. The integration requires that iView ETG is version 2.7.1.

Note: Earlier or later versions of iView ETG are not compatible.

On the QTM computer it is required that you install the Qualisys SMI server. You can download the SMI com server by following the **Download device drivers** link on the **Input Devices** page in **Project options**.

The only connection that is needed for SMI is that the two computers must be on the same network. Check with your IT administrator if you don't know how to achieve this.

Add SMI eye tracker to a project

The SMI eye tracker can be captured together with all the other data in QTM. Follow these instructions to add SMI to your project:

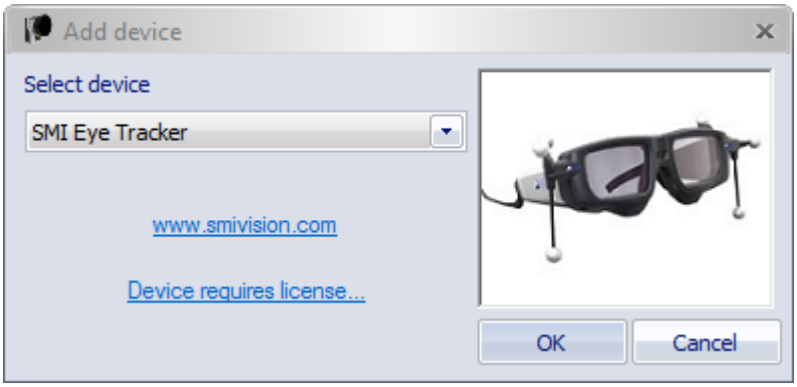
1. Make sure that the SMI computer is connected on the same network as the QTM computer and that you installed the SMI com server, see chapter “SMI eye tracker installation” on the previous page.

   Note: The SMI eye tracker has no synchronization options so you do not need to connect any other signals. This means that data from SMI is only synchronized via a start command, but you can modify the delay between SMI and the marker data in QTM to find the best option.

2. Start the SMI iView ETG program.

   a. For more information about the iView ETG program and for example how to track the eye the best way please contact SMI.

3. To add the SMI eye tracker device, open the **Input Devices** page in the **Project options** dialog and click on **Add device**.



   a. Select the **SMI Eye Tracker** from the list with devices and click OK.

   Note: QTM requires the **EyeTracker** plugin to capture data with the SMI eye tracker. Contact Qualisys AB if you want to buy the plugin.

4. Make sure that the SMI device is enabled in the list. Then double-click on the **SMI** line in the list to open the **SMI** page.

5. On the **SMI** page enter the following information.

   IMAGE

   Address
   The IP address or name of the SMI computer.

   Data offset
   The offset of the SMI data compared to the marker data. You can test this by capturing a subject looking at a moving marker. After that you will need to reprocess the file with different data offsets and find out where the gaze diverges a little as possible from the marker. One way to

see this is to turn on the Gaze vector trace on the **3D View Settings** page in Project options.

6. Go to the **Processing** page and turn on the **Calculate 6DOF** and **Calculate gaze vector data** option for both **Real time actions** and **Capture actions**
7. Go to the **Gaze Vector** page to add a gaze vector.
   a. Click on **Add** to add a new gaze vector. The gaze vector includes the definition for both left and right eye.
   b. Double click on the gaze vector to open the Gaze vector dialog.

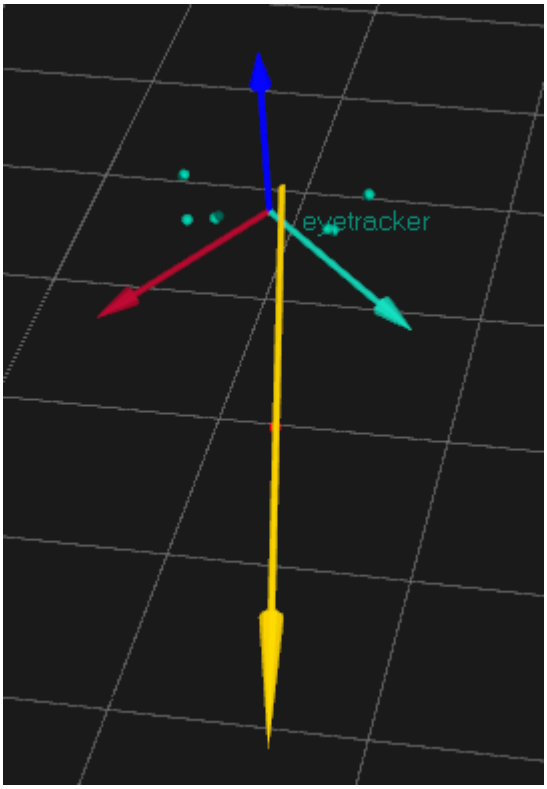
      IMAGE
   c. Select **SMI** on the **Eye Tracker** option.
   d. Select the rigid body that you use for the eye tracker on the **Rigid body** option, see information in the next step.
8. You need to make sure that you have a rigid body on the SMI eye tracker and that it is defined in QTM. There are pre-built rigid bodies from SMI that are included in a kit from SMI. If you don't have the kit then you need to make your own rigid body.
   a. Pre-built rigid body
      i. When you add the gaze vector then the pre-built rigid bodies are created automatically in the rigid bodies list on the **6DOF Tracking** page in the **Project Options** dialog.
      ii. The two pre-defined bodies are called, SMI ETG Layout 4 and SMI ETG Layout 5.
      iii. Test in QTM which body that you have on your SMI glasses.
   b. Own rigid body
      i. Put at least 4 markers on the eye tracker, it is good if the markers are on both sides of the eye tracker.
      ii. Place the eye tracker with the rigid body in the measurement volume. It is recommended that you have it on a person for the best definition.
      iii. Make a short measurement, about 3 seconds, with the person standing still looking straight forward.
      iv. Select the markers of the rigid body. Define the body by right-clicking on them and selecting **Define rigid body (6DOF)**. You can give it any name you want.
      v. Start preview with **New** on the **File** menu.
      vi. Go to the **6DOF tracking** page.
      vii. Then translate the local coordinate system of the rigid body. It has to be translated to the position of the video camera at the bridge of the glasses.
      viii. You then need to rotate the coordinate system with the **Rotate** option so that Z is pointing forwards and Y is pointing upwards.
      ix. If needed, correct the rotation by comparing the gaze vector to the position of a visible target marker.

Making a measurement with SMI

The eye tracker data is collected together with the marker data in QTM and can be reprocessed and exported together with the other data, see chapter “Process and export SMI gaze vector data” below. Follow these steps to capture eye tracker data.

1. Enable the SMI eye tracker on the **Input devices** page and add a gaze vector on the **Gaze Vector** page, see chapter “Add SMI eye tracker to a project” on page E - 16.
2. Start preview in QTM and the gaze vector is displayed in the 3D view window as soon as the 6DOF body is identified.

   Note: Make sure that you have activated **Calculate gaze vector data** and **Calculate 6DOF** on the **Processing** page in **Project options**.



3. You can also view the Gaze vector data in the **Data info** window, see chapter “SMI data in QTM” on page E - 20.
4. The eye-tracking data is captured automatically with the other data in QTM. Since there is no trigger or sync connection on the SMI the capture is started via software. This means that the synchronization cannot be known for sure. However the delay can be specified in the settings.
5. The frequency of the SMI eye tracker is either 30 Hz or 60 Hz, which is detected automatically by QTM. The lower frequency means that the eye-tracker data will update less often than the marker data. So it might look like it is more out sync than it actually is.

Process and export SMI gaze vector data

The Gaze vector data can be reprocessed in a file, both in reprocessing and batch processing. This is useful if you need to update the gaze vector data because of changed

6DOF data.

If you have changed the 6DOF data then you need to reprocess the Gaze vector data to update it. Follow these steps to reprocess it.

1. Click on **Reprocess** on the **Capture** menu.
2. Then select **Calculate gaze vector data** in the **Processing steps**.
   a. If you need to change the rigid body that is associated with the gaze vector, then go to the **Gaze vector** page and double click on the gaze vector in the list.
3. Click **OK** in the File reprocessing dialog.

The Gaze vector data can then be exported and analyzed in external programs. You can either export it to TSV or MAT files.

TSV
To export the eye tracker data select the **Export eye tracker data if available** option for the TSV export. The result is two TSV file, *_g_1.tsv and *_g_2.tsv with gaze vector data for left respectively right eye. The two files includes the following data:

_g.tsv

GAZE_VECTOR_NAME
The name of the gaze vector.

NO_OF_SAMPLES
The number of samples in the gaze vector data.

FREQUENCY
The frequency of the gaze vector data.

TIME_STAMP
Date and time when the motion capture file was made. The date and time is followed by a tab character and then the timestamp in seconds from when the computer was started.

WARNING: The time stamps indicate the start of the capture according to the QTM software. The time stamps may not exactly correspond to the first frame of the capture and it is discouraged to use them for synchronization with data from other devices connected to the same computer.

START_OFFSET
The offset time for the first gaze vector frame. It is needed since you may not always cut the measurement range at the start of gaze vector frame. Reduce the start time of the gaze vector data with the offset so that it actually starts before the marker data.

Gaze vector data
Columns with the gaze vector data in the following order: X Pos, Y Pos, Z Pos, X Vec, Y Vec, Z Vec

MAT
To export the eye tracker data select the **Eye tracker** option for the MATLAB file export. The struct array of the MAT file then includes the following data:

GazeVector
The struct array consists of the following elements for each eye. For example write GazeVector(1).GazeVectorName to access the name for the left eye.

GazeVectorName
The name of the gaze vector.

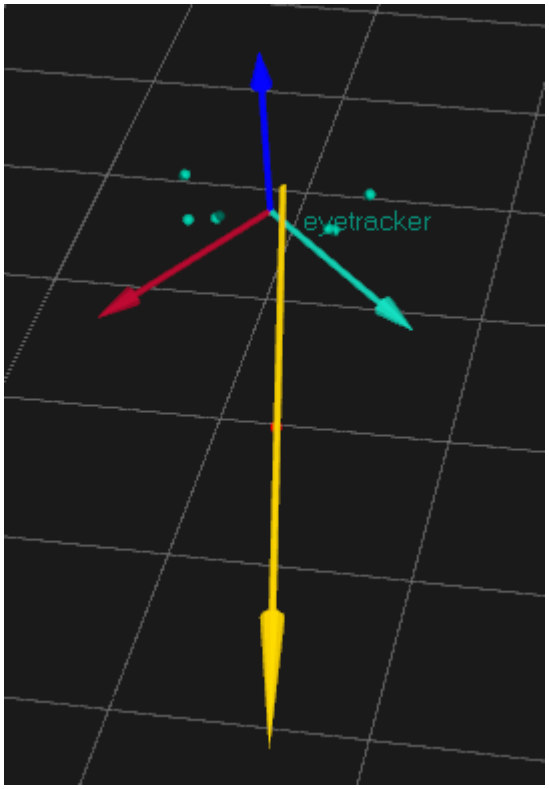
NrOfSamples
The number of samples in the gaze vector data.

StartOffset
The offset time for the first gaze vector frame. It is needed since you may not always cut the measurement range at the start of gaze vector frame. Reduce the start time of the gaze vector data with the offset so that it actually starts before the marker data.

Frequency
The frequency of the gaze vector data.

GazeVector
Array with the gaze vector data in the following order: X Pos, Y Pos, Z Pos, X Vec, Y Vec, Z Vec

SMI data in QTM



The Gaze vector data is displayed in the 3D view of the window as a vector, for example color and length can be modified on the **3D view settings** page in **Project options**.

The SMI eye tracker data in QTM consists of just the gaze vector data. The gaze vector is calculated by the SMI iView ETG program and then in QTM the vector is positioned at the correct position in reference to the rigid body.

Gaze Vector Data

Device	X Pos	Y Pos	Z Pos	X Vec	Y Vec	Z Vec	Frame Number
Gaze vector 1 (L)	533.95	778.60	1488.75	0.12	0.98	-0.16	1
Gaze vector 1 (R)	595.02	777.64	1486.61	-0.08	0.99	0.09	1

Gaze vector data
The gaze vector data includes:

Device
The device is the name of the Gaze vector. (L) and (R) stands for left and right eye.

X Pos
This is the X position for the origin of the Gaze vector in the Global coordinate system.

Y Pos
This is the Y position for the origin of the Gaze vector in the Global coordinate system.

Z Pos
This is the Z position for the origin of the Gaze vector in the Global coordinate system.

X Vec
This is the X value of the Gaze vector in the Global coordinate system.

Y Vec
This is the Y value of the Gaze vector in the Global coordinate system.

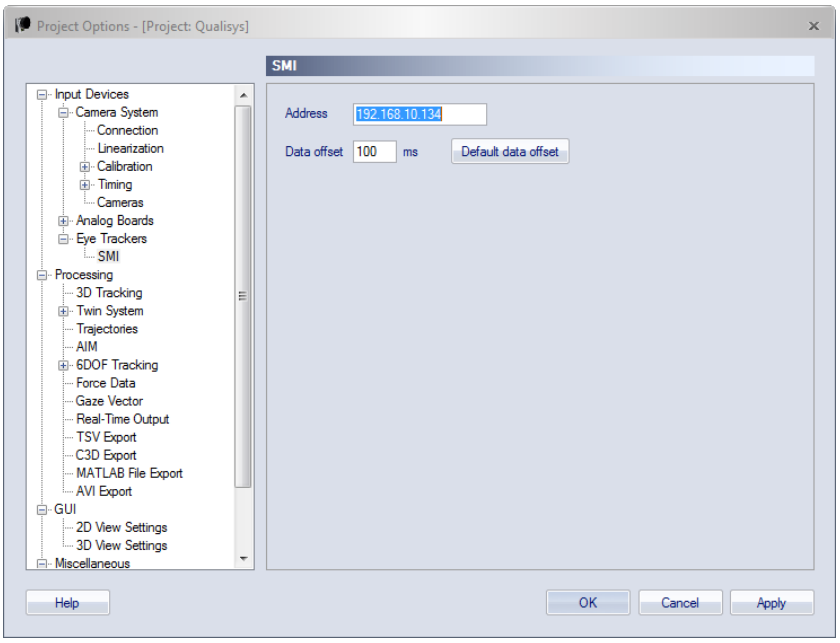
Z Vec
This is the Z value of the Gaze vector in the Global coordinate system.

Frame Number
This is the frame number of the SMI data. Since the frame rate of the SMI data is lower than the marker data, it means that the SMI data is updated less often than the marker data.

SMI QTM settings

The settings for SMI eye tracker is split on two different pages in **Project Options**:**SMI** and **Gaze Vector**. The SMI page controls the communication with the SMI computer. The Gaze Vector page on the other and control the calculation of gaze vector data.

SMI



The SMI page contains the settings for communication with SMI iView program.

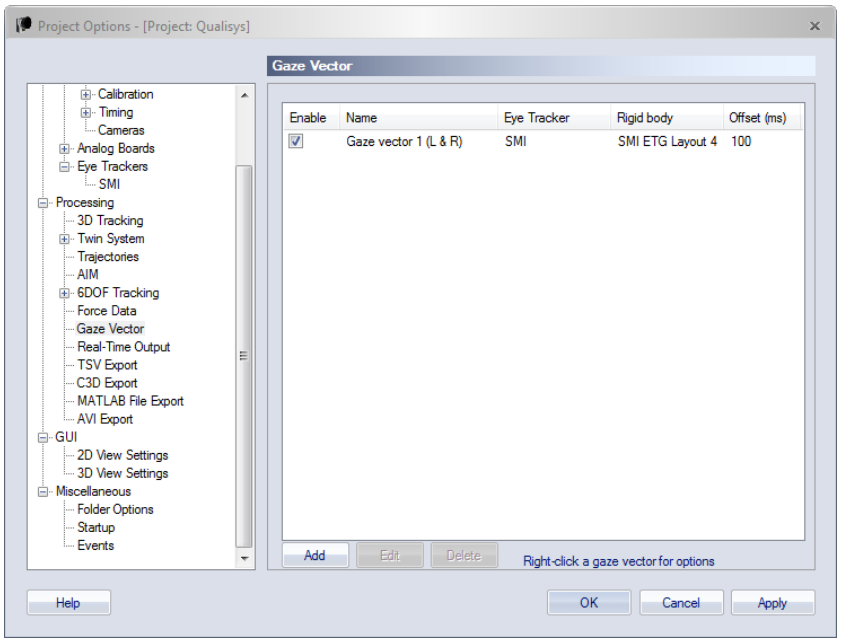
Address
Enter the IP address of the computer with the SMI iView program.

Data offset
Enter the data offset of the eye-tracker data. The default is 100 ms. You can for example measure the offset by looking at a moving marker and get as little difference between the Gaze vector and the marker position.

Default data offset
Set the **Data offset** option to the default 100 ms.

You will also need to create a Gaze vector to capture the eye-tracker data, see chapter “Gaze Vector” on the facing page

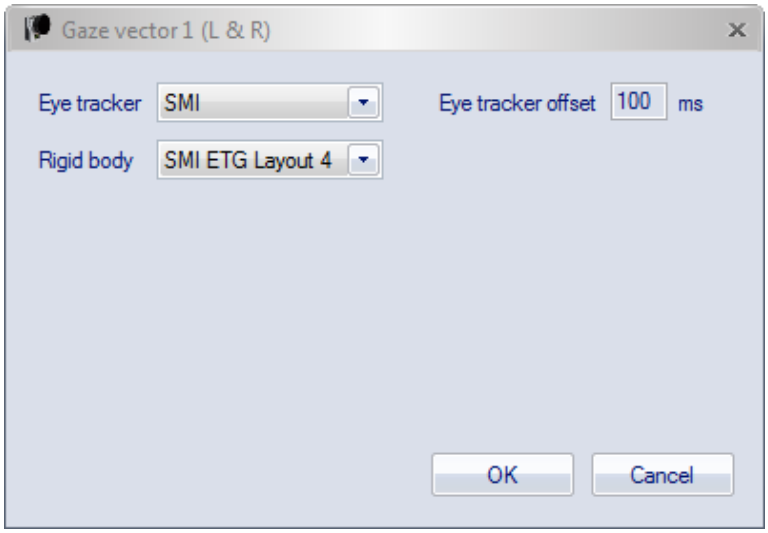
Gaze Vector



The **Gaze Vector** page contains a list of the current gaze vectors and the settings needed to calculate a gaze vector from the eye tracker data. Use the following options to modify the list.

Add
Add a new gaze vector to the list.

Edit (or double-click on a gaze vector)
These two alternatives open the dialog below for the selected gaze vector.



The options in the dialog defines how to calculate the Gaze vector.

Eye tracker
Select an eye tracker from the list.

In **Project options** the list contains all of the eye trackers that are available on the **Input devices** page. For reprocessing with file settings the list contains all of the eye trackers that were used in the capture.

Eye tracker offset
The **Eye tracker offset** is the number of milliseconds that the eye tracker data

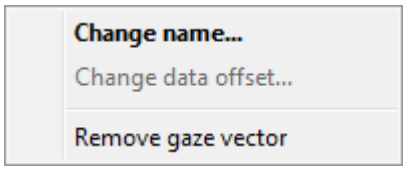
is delayed compared with the marker data. This value is then used to compensate to align the data in the file.

Rigid body
Select the rigid body that is mounted on the eye tracker. This is used as reference for the gaze vector calibration. There are two pre-defined rigid bodies for SMI ETG called **SMI ETG Layout 4** and **SMI ETG Layout 5**.

Note: In **Project options** the list contains all of the rigid bodies that are available on the **Input devices** page. For reprocessing with file settings the list contains all of the eye trackers that were used in the capture.

Delete
Delete the selected gaze vector.

Right-click to open menu



From the menu you can **Change name** of the selected gaze vector and **Remove gaze vector**.

Tobii eye tracker

The Tobii Pro Glasses 2 eye tracker is integrated with QTM so that you can view and export gaze vector data. For the integration a Qualisys-Tobii connectivity kit is available, containing:

- An attachment for 6DOF tracking of the glasses
- A cable for hardware synchronization

The connectivity kit can be purchased from Tobii or via Qualisys AB.

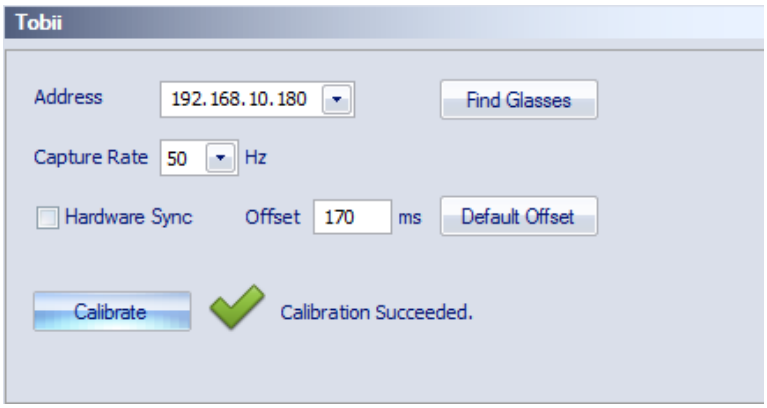
For information on how to set up a Tobii eye tracker in QTM, see the following chapters.

Add Tobii eye tracker to a project

The Tobii eye tracker can be captured together with all the other data in QTM. For the connection, the QTM computer and the Tobii Recorder should be connected to the same local network. The Tobii Recorder should be connected via WiFi. Follow these instructions to add Tobii to your project:

1. Configure the Tobii glasses network connection (WiFi) using the Tobii Pro Glasses Controller software.
2. To add the Tobii eye tracker device, open the **Input Devices** page in the **Project options** dialog and click on **Add device**.

3. Select the **Tobii Pro Glasses 2** from the list with devices and click **OK**.
   Note: QTM requires the EyeTracker plugin to capture data with the Tobii eye tracker. Contact Qualisys AB if you want to buy the plugin.
4. Make sure that the Tobii device is enabled in the list. Then double-click on the **Tobii** line in the list to open the **Tobii** page.
5. On the **Tobii** page enter the following information.



Address
The IP address or name of the Tobii glasses. Press “Find Glasses” to find them on the network and update the address field.

Capture rate
Select capture rate. Note that not all versions of the glasses support 100 Hz.

Hardware Sync
Enable synchronized capture start between QTM and Tobii glasses. See chapter “Setting up Tobii hardware synchronization” on the next page for how to set up hardware synchronization.

Offset
The offset of the Tobii data compared to the marker data. You can test this by capturing a subject looking at a moving marker. After that you will need to reprocess the file with different data offsets and find out where the gaze diverges a little as possible from the marker. One way to see this is to turn on the Gaze vector trace on the **3D View Settings** page in Project options. Press “Default Offset” to reset the offset to default value.

Calibrate
Initiate Tobii calibration. Calibration is done the same way as in the Tobii Pro Glasses Controller. By looking at a card with a marker on.

6. Go to the **Processing** page and turn on the **Calculate 6DOF** and **Calculate gaze vector data** option for both **Real time actions** and **Capture actions**.
7. Go to the **Gaze Vector** page to add a gaze vector. For an overview of the gaze vector options, see “Gaze vector settings” on page E - 33.
   a. Click on **Add** to add a new gaze vector. The gaze vector includes the definition for both left and right eye.
   b. Double click on the gaze vector to open the Gaze vector dialog.
   c. Select **Tobii** on the **Eye Tracker** option.
   d. Select the rigid body that you use for the eye tracker on the **Rigid body** option, see information in the next step.
   e. Optionally, check the **Use median filter** option for smoothing the gaze position and gaze vector data.
8. You need to make sure that you have a marker extension (rigid body) on the Tobii glasses and that it is defined in QTM. There are pre-built rigid bodies available from Tobii. When you add the gaze vector, the pre-built rigid bodies are created automatically in the rigid bodies list on the **6DOF Tracking** page in the **Project Options** dialog. The pre-defined bodies are called, **Tobii Layout 1**, etc. If you are not sure which layout you have you can test in QTM which definition fits best to the marker extension on your Tobii glasses.

   Note: For the best tracking of the glasses, it is highly recommended to refine the 6DOF definition as described in chapter “Refinement of Tobii rigid body definition” on the facing page. When not using a refined rigid body definition, you may need to increase the **Bone length tolerance** to a value of 10 mm.

Setting up Tobii hardware synchronization

For hardware synchronization of the Tobii data in QTM, you can use the synchronization cable that is included in the Tobii-Qualisys connectivity kit.

For **Arqus** and **Miqus** systems a **Camera Sync Unit** is required for hardware synchronization.

1. Use a BNC cable to connect the Tobii sync cable to the **MEAS. TIME** port on the **Camera Sync Unit**.
2. Connect the 3.5" connector of the Tobii sync cable to the **Data synchronization** port on the Tobii recorder.

For **Oqus** systems with a Trigger/Sync splitter:

1. Connect a Trigger/Sync splitter to the control port of one of the cameras.
2. Use a BNC cable to connect the Tobii sync cable to **Sync out** on the splitter cable.
3. Connect the 3.5" connector of the Tobii sync cable to the **Data synchronization** port on the Tobii recorder.
4. Do the following settings in QTM:
   a. Open the **Synchronization** page under **Project Options**.
   b. Select the Oqus camera which is used for the synchronization.
   c. Set **Mode** under **Synchronization output** to **Measurement time**.

Note: The synchronization connection is only needed at the start of the capture. After starting the capture, the synchronization cable may be disconnected.

Note: Hardware synchronization and offset compensation is only applied to captured data. It is not possible to compensate for latency in real time.

Refinement of Tobii rigid body definition

QTM contains standard 6DOF definitions of two standard rigid body lay outs for Tobii glasses. However, the actual 6DOF lay outs may slightly deviate. Therefore, for optimal tracking of the Tobii glasses, it is recommended to refine the rigid body definition. This can be done following the steps below.

1. Make a short capture with the Tobii glasses with the original rigid body definition (Tobii Lay out 1 or 2). In the 6DOF settings, use a larger bone length tolerance of 10-15 mm.
2. Select a range on the time line with good quality tracking and make sure that the Tobii lay out trajectories are correctly labeled. 6DOF data should be present. Check that the local axes are correct: the Z axis should point forward from the glasses and the Y axis should point up.
3. Select the six Tobii lay out trajectories, right click the selection, and choose **Define rigid body (6DOF)** (choose **Average of frames** if you have checked the tracking and labeling quality across the selected time range). Give the rigid body a different name (for example "Tobii-refined").

After adding the new definition, you will need to correct the definition as follows.

1. Open the **6DOF Tracking** page under **Project Options**.
2. Select the new rigid body (from now on called "Tobii-refined")
3. Press the **Translate** button, choose the option **To current position of this rigid body**, and select the original Tobii lay out. Press **OK** twice (Translate dialog and Project Options).
4. Reprocess the file, make sure to check the **Calculate 6DOF** option and select 6DOF settings from **project**.
5. Go back to the **6DOF Tracking** page under **Project Options** and select the Tobii-refined rigid body.
6. Press the **Rotate** button, choose the option **Rotate as this rigid body**, and select the original Tobii lay out. Press **OK** twice (Rotate dialog and Project Options).
7. Reprocess the file, make sure to check the **Calculate 6DOF** option and select 6DOF settings from **project**.
8. Check in the 3D View window that the local axes of the two rigid bodies perfectly overlap.

Now you have a refined rigid body definition, make sure it is used for the eye tracker.

1. Open the **Gaze Vector** page under **Project Options** and select it as the rigid body for your Tobii eye tracker.
2. Open the **6DOF Tracking** page under **Project Options**, set the **Bone length tolerance** value to 5 mm, and remove the unused rigid body definitions.

Making a measurement with Tobii

The eye tracker data is collected together with the marker data in QTM and can be reprocessed and exported together with the other data, see chapter " Process and export Tobii gaze vector data" below. Follow these steps to capture eye tracker data in QTM.

1. Enable the Tobii eye tracker on the Input devices page and add a gaze vector on the Gaze Vector page, see chapter "Add Tobii eye tracker to a project" on page E - 24.
2. Make sure that you have activated **Calculate gaze vector data** and **Calculate 6DOF** on the **Processing page** in **Project options**.
3. Start preview in QTM and the gaze vector is displayed in the 3D view window as soon as the 6DOF body is identified.
4. You can also view the gaze vector data in the **Data info window**, see chapter "Tobii data in QTM" on page E - 31.
5. The eye-tracking data is captured automatically with the other data in QTM.
   a. When not using hardware synchronization the capture is started via software. This means that the synchronization cannot be known for sure. However, the latency can be compensated for by specifying a representative offset value in the settings (see **Offset** in "Add Tobii eye tracker to a project" on page E - 24).
   b. When using hardware synchronization, the capture is started simultaneously with the mocap data via a sync cable. For instruction on how to set up hardware synchronization, see chapter "Setting up Tobii hardware synchronization" on page E - 26.
6. The frequency of the Tobii eye tracker is either 50 Hz or 100 Hz. The lower frequency means that the eye-tracker data will update less often than the marker data. So it might look like it is more out sync than it actually is.

Process and export Tobii gaze vector data

The Gaze vector data can be reprocessed in a file, both in reprocessing and batch processing. This is useful if you need to update the gaze vector data because of changed 6DOF data.

If you have changed the 6DOF data then you need to reprocess the Gaze vector data to update it. Follow these steps to reprocess it.

1. Click on **Reprocess** on the **Capture** menu.
2. Then select **Calculate gaze vectordata** in the **Processing steps**.
   a. If you need to change the rigid body that is associated with the gaze vector, then go to the **Gaze vector** page and double click on the gaze vector in the list.
3. Click **OK** in the File reprocessing dialog.

The gaze vector and eye tracker data can then be exported and analyzed in external programs. You can either export it to TSV or MAT files.

TSV
To export the eye tracker data select the data type **Eye tracker** in the TSV export options. The gaze vector and eye tracker data will be exported in separate files,

for example *_g_1.tsv and *_g_2.tsv with gaze vector data for the left and right eye, respectively, and *_e_1.tsv with eye tracker data. The data included in the gaze vector export is:

_g.tsv

GAZE_VECTOR_NAME
The name of the gaze vector.

NO_OF_SAMPLES
The number of samples in the gaze vector data.

FREQUENCY
The frequency of the gaze vector data.

TIME_STAMP
Date and time when the motion capture file was made. The date and time is followed by a tab character and then the timestamp in seconds from when the computer was started.

START_OFFSET
The offset time for the first gaze vector frame. It is needed since you may not always cut the measurement range at the start of gaze vector frame. Reduce the start time of the gaze vector data with the offset so that it actually starts before the marker data.

HW_SYNC
Indicates if hardware sync was used (value: **YES**) for the eye tracker data or not (value: **NO**).

FILTER
Indicates if the gaze vector data was filtered (value: **YES**) or not (value: **NO**).

Gaze vector data
Columns with the gaze vector data in the following order: X Pos, Y Pos, Z Pos, X Vec, Y Vec, Z Vec.

The data included in the eye tracker data export is:

_e.tsv

EYE_TRACKER_NAME
The name of the eye tracker device.

NO_OF_SAMPLES
The number of samples in the eye tracker data.

FREQUENCY
The frequency of the eye tracker data.

TIME_STAMP
Date and time when the motion capture file was made. The date and time is followed by a tab character and then the timestamp in seconds from when the computer was started.

START_OFFSET
The offset time for the first eye tracker data frame.

HW_SYNC
Indicates if hardware sync was used (value: **YES**) for the eye tracker data or not (value: **NO**).

Gaze vector data
Columns with the gaze vector data in the following order: X Pos, Y Pos, Z Pos, X Vec, Y Vec, Z Vec.

MAT
To export the eye tracker data select the **Eye tracker** option for the MATLAB file export. The struct array of the MAT file then includes the following data:

GazeVector
The struct array consists of the following elements for each eye. For example write GazeVector(1).GazeVectorName to access the name for the left eye.

GazeVectorName
The name of the gaze vector.

NrOfSamples
The number of samples in the gaze vector data.

StartOffset
The offset time for the first gaze vector frame. It is needed since you may not always cut the measurement range at the start of gaze vector frame. Reduce the start time of the gaze vector data with the offset so that it actually starts before the marker data.

Frequency
The frequency of the gaze vector data.

HWSync
Indicates if hardware sync was used for the eye tracker data or not. The values can be 0 (no hardware sync) or 1 (hardware sync).

Filter
Indicates if the gaze vector data was filtered or not. The values can be 0 (not filtered) or 1 (filtered).

GazeVector
Array with the gaze vector data in the following order: X Pos, Y Pos, Z Pos, X Vec, Y Vec, Z Vec.

EyeTracker
Struct array with the following eye tracker data.

EyeTrackerName
The name of the eye tracker device.

NrOfSamples
The number of samples of eye tracker data.

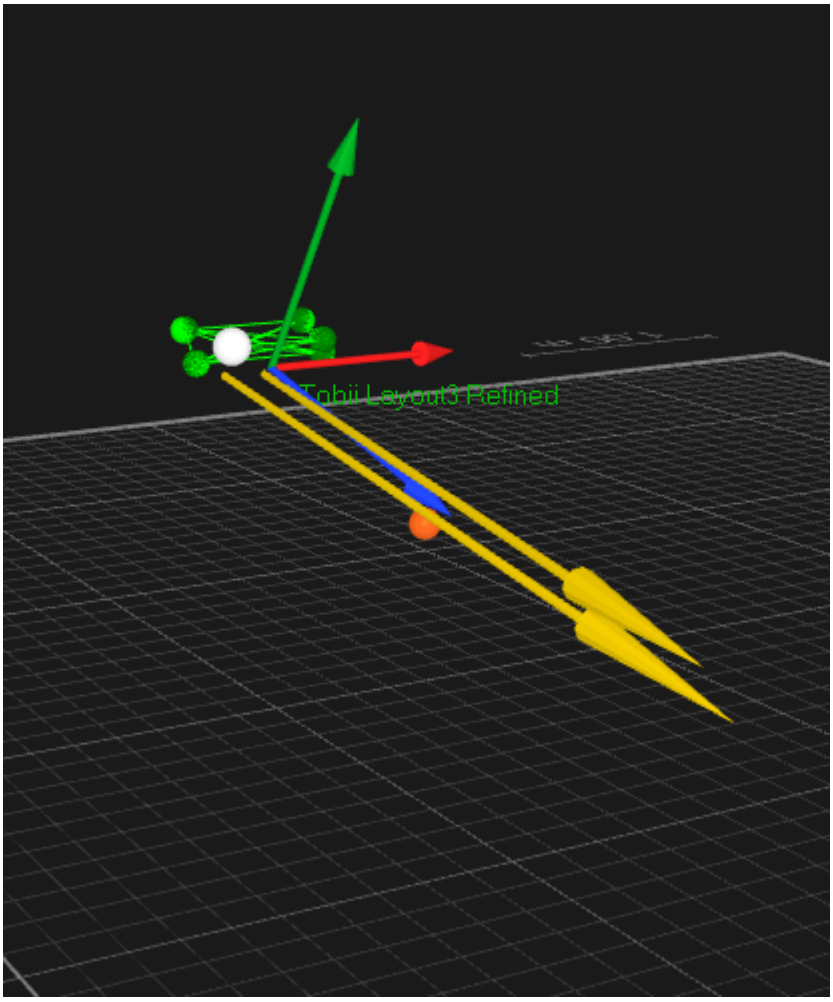
StartOffset
The offset time for the first eye tracker data frame.

Frequency
The frequency of the eye tracker data.

HWSync
Indicates if hardware sync was used for the eye tracker data or not. The values can be 0 (no hardware sync) or 1 (hardware sync).

EyeTracker
Array with the pupil diameter data in mm in the following order: left pupil, right pupil.

Tobii data in QTM



The Gaze vector data is displayed in the 3D view of the window as a vector, for example color and length can be modified on the **3D view settings** page in **Project options**.

There are two types of data for Tobii eye trackers in QTM, namely eye-tracker data and gaze vector data. The gaze vector is provided by the Tobii recorder and then transformed in QTM to global 3D coordinates through the pose of the associated rigid body.

Gaze Vector Data

Device	X Pos	Y Pos	Z Pos	X Vec	Y Vec	Z Vec	Frame Number
Gaze vector 1 (R)	151.15	68.59	1238.74	-0.17	0.98	-0.06	316
Gaze vector 1 (L)	88.88	69.11	1231.33	-0.06	1.00	-0.06	316

Gaze Vector data
The gaze vector data includes:

Device
The device is the name of the Gaze vector. (L) and (R) stands for left and right eye.

X Pos
This is the X position (mm) for the origin of the Gaze vector in the Global coordinate system.

Y Pos
This is the Y position (mm) for the origin of the Gaze vector in the Global coordinate system.

Z Pos
This is the Z position (mm) for the origin of the Gaze vector in the Global coordinate system.

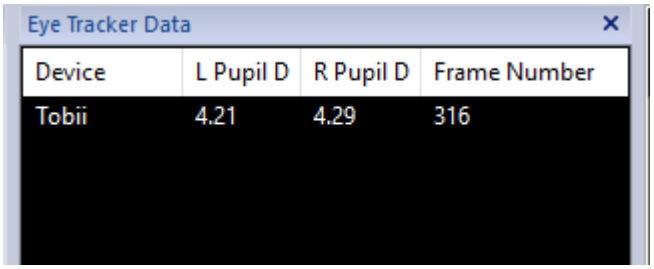
X Vec
This is the X value of the Gaze vector in the Global coordinate system.

Y Vec
This is the Y value of the Gaze vector in the Global coordinate system.

Z Vec
This is the Z value of the Gaze vector in the Global coordinate system.

Frame Number
This is the frame number of the Tobii data. Since the frame rate of the Tobii data is mostly lower than the marker data, it means that the Tobii data is updated less often than the marker data.

The eye-tracker data is provided by the Tobii recorder.



Eye Tracker data
The Tobii eye tracker data includes:

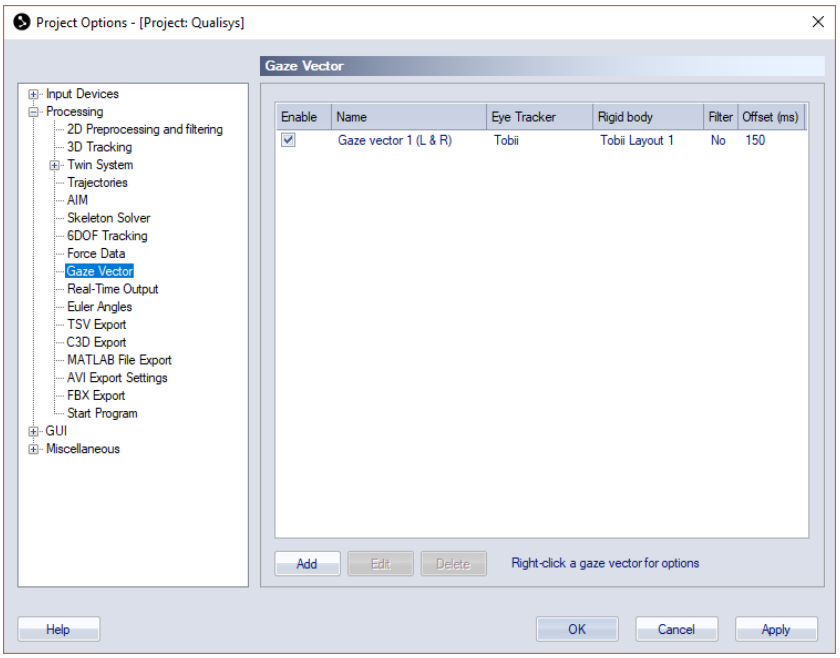
Device
The name of the Tobii eye tracker device under **Input devices**. The name for a single Tobii device is by default "Tobii". Additional Tobii devices are numbered with 2, 3, etc.

L Pupil D
The diameter of the left pupil in mm.

R Pupil D
The diameter of the right pupil in mm.

Frame Number
Frame number of the Tobii data. Note that the Tobii frame number is not the same as the QTM frame number.

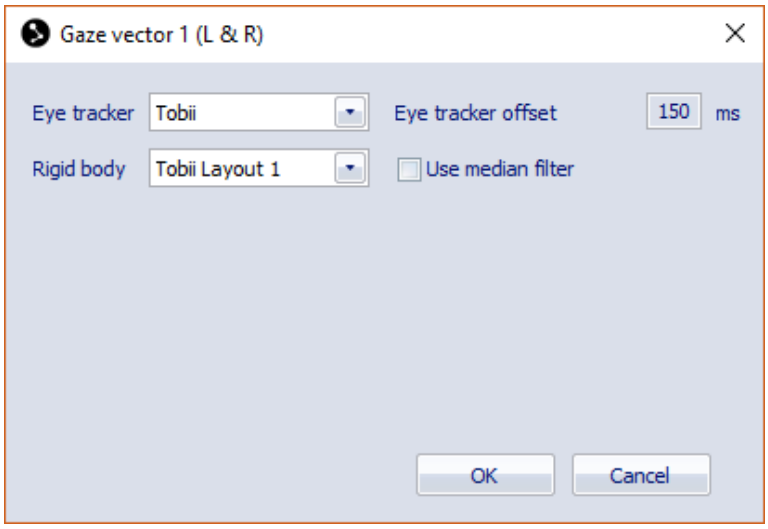
Gaze vector settings



The **Gaze Vector** page contains a list of the current gaze vectors and the settings needed to calculate a gaze vector from the eye tracker data. Use the following options to modify the list.

Add
Add a new gaze vector to the list.

Edit (or double-click on a gaze vector)
These two alternatives open the dialog below for the selected gaze vector.



The options in the dialog defines how to calculate the Gaze vector.

Eye tracker
Select an eye tracker from the list.

In **Project options** the list contains all of the eye trackers that are available on the **Input devices** page. For reprocessing with file settings the list contains all of the eye trackers that were used in the capture.

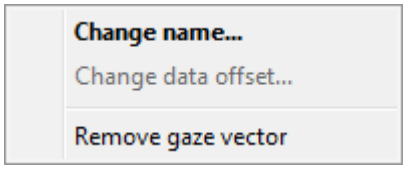
Eye tracker offset
The **Eye tracker offset** is the number of milliseconds that the eye tracker data is delayed compared with the marker data. This value is then used to compensate to align the data in the file.

Rigid body
Select the rigid body that is mounted on the eye tracker. This is used as reference for the gaze vector calibration.
Note: In **Project options** the list contains all of the rigid bodies that are available on the **Input devices** page. For reprocessing with file settings the list contains all of the eye trackers that were used in the capture.

Use median filter
Check this option for applying a median filter for smoothing of the gaze position and gaze vector data.

Delete
Delete the selected gaze vector.

Right-click to open menu



From the menu you can **Change name** of the selected gaze vector and **Remove gaze vector**.

ASL EyeHead integration

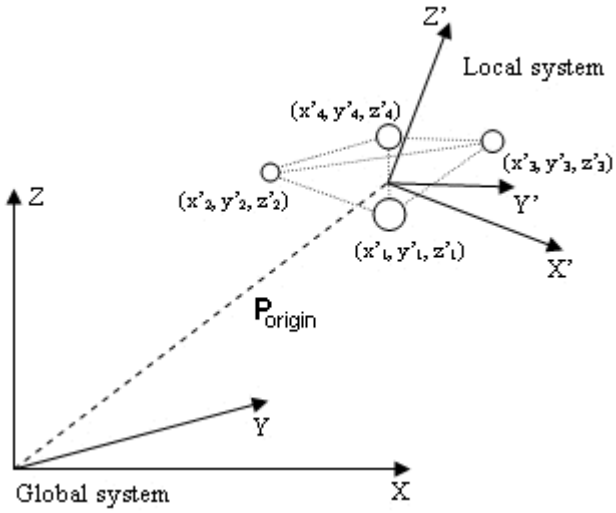
The ASL EyeHead Integration system combines information from the head mounted eye tracker with information from the camera system about the heat position to compute line of gaze in the environment. The head mounted eye tracker determines line of gaze relative to the head, and the camera system determines the head position and orientation of the head in the environment.

For the integration you need one rigid body on the head of the subject and then a rigid body on a tool that measures the position of the eye. Finally you need to define a plane with a laser pointer or a pointer wand. All of this equipment can be ordered from ASL. The data from the rigid bodies are then used to calculate the gaze in the Mobile Eye EyeVison program.

For more information about the integration please contact ASL.

Appendix F: 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:

$$\mathbf{P}_{global} = \mathbf{R} \cdot \mathbf{P}_{local} + \mathbf{P}_{origin}$$

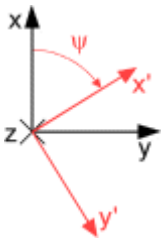
If the 6DOF data refer to another coordinate system than the global coordinate the position and rotation is calculated in reference to that coordinate system instead. The coordinate system for rigid body data is then referred to the global coordinate system.

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 (**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 G: General PAF information

Project Automation Framework (PAF)

Project Automation Framework (PAF) is a plugin for QTM that automates measurement and analysis in for example Visual3D or Matlab. In QTM 2.17 and later, PAF is included, allowing you to create your own customized modules for data collection and analysis. For more information about PAF, please refer to the Project Automation Framework Manual (PDF) in the Packages folder of the QTM installation.

In addition, predefined analysis modules are available, for example for Gait, Running and Baseball. For more information about the available analysis modules, check our website or contact your local distributor or sales@qualisys.com.

The following chapters describes the parts that are generic for all PAF modules.

PAF module installation

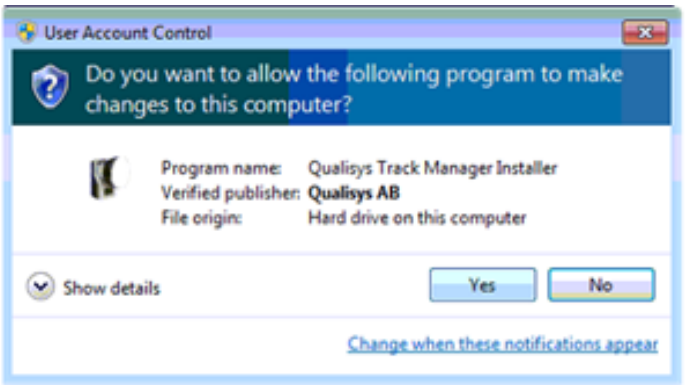
Downloading installation files

The required files are available in the Qualisys client login, see http://www.qualisys.com/my/. Log in with your QTM user name and your licence key as password. You can find this information in QTM under Help > About.

New installation

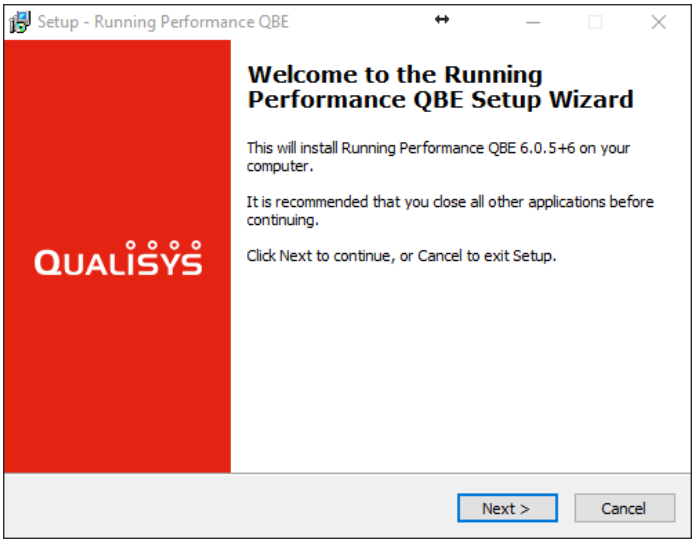
Follow these instructions if you are installing the module for the first time. If you are updating an existing installation, see next section.

1. Start QTM. Open any project.
2. Either compressed installer (*.qpi) or executable installer (*.exe) can be used to install the module.
    a. If installing the module from qpi file, double-click the *.qpi file that you downloaded from the Qualisys client login. Allow QTM to make changes if Windows prompts you to do so:



After 10-15 seconds QTM will show a “Success” message.

b. If installing the module from executable file, click the executable *.exe file that you downloaded from the Qualisys client login. Follow the instruction.



3. Create a new project. Use this project to collect all future data that you want to analyze with the PAF module:

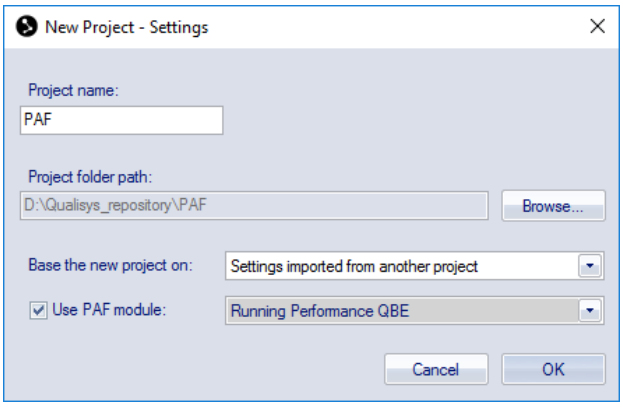
   a. Select File > New Project.

   b. Enter name for the project, for example “PAF Project”. Change the location of the project folder if you wish to, however we recommend that you keep it in Documents unless multiple different users need access.

   c. If you collected data previously and you want to use the same camera settings, choose “Settings imported from another projects” and select a project that contains the correct camera settings that you want to use. Otherwise select “Use default settings”.

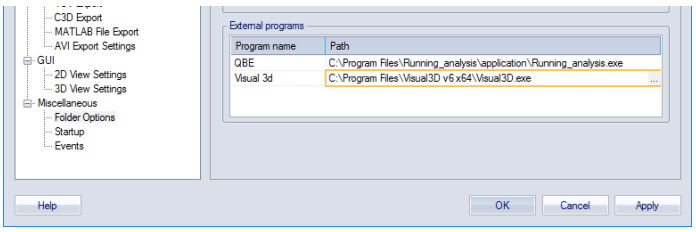
   d. Check “Use PAF module” and select the correct module from the drop-

down.



4. If you selected to import the settings from another project, choose the settings file in the next dialog.
5. Enter the module license you received from Qualisys when prompted to do so. Select the name of the module in the Plug-in drop-down. For Gait modules, always select PAF Gait Package, regardless of the exact name of the module you are using:

6. Open the Project Options, select Folder Options and specify the location of the computational engine. If using Visual3D, locate Visual3D.exe by clicking the "…" icon (on most computers the path is 'C:\Program Files\Visual3D v6 x64'). If using Reckon3D locate Reckon3D.exe at 'C:\Program Files (x86)\Reckon3D'. If using QBE navigate to' C:\Program Files\Running_analysis\application':



7. Check the C3D export settings in Project Options match the screenshot below:

   Check Exclude unidentified trajectories

   Check Exclude empty trajectories

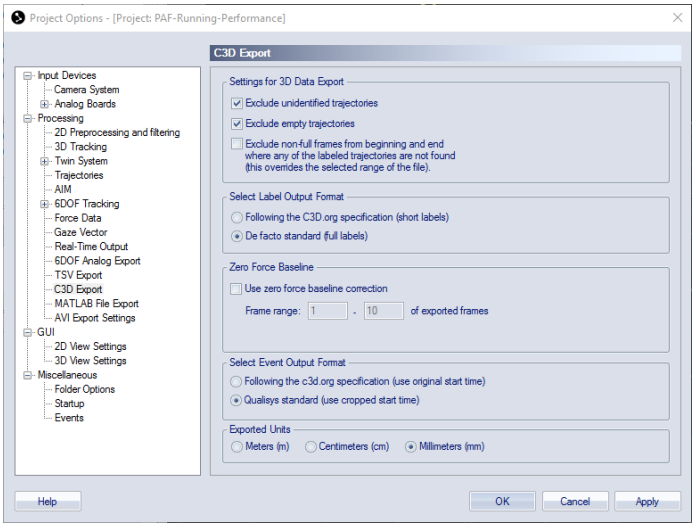
   Un-check Exclude non-full frames

   Choose De facto standard

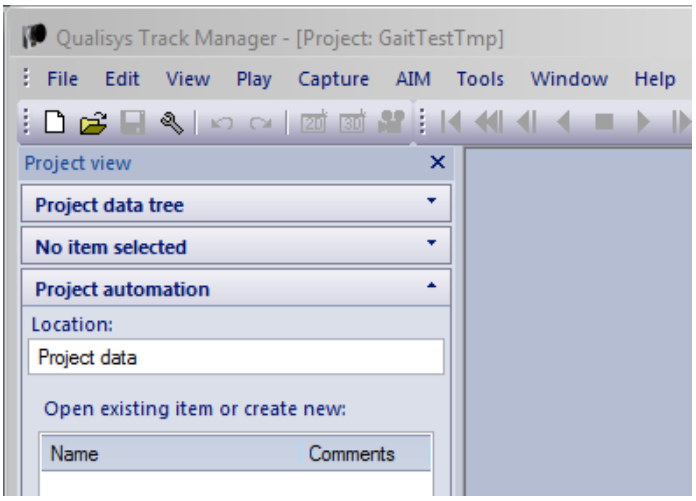
Export units are millimeters

Zero force baseline can be used if the force plate is unloaded at the beginning of the measurement but note that depending on which module you are using the Visual3D/Reckon3D script is set up to apply a zero baseline correction as well. The Visual3D/Reckon3D correction will overwrite the zero frame range that is set here.

Note that physically zeroing the force plates is also required prior to data collection if the amplifier has a zero button.



8. Open the Project View by selecting View > Project View (shortcut Ctrl+R).

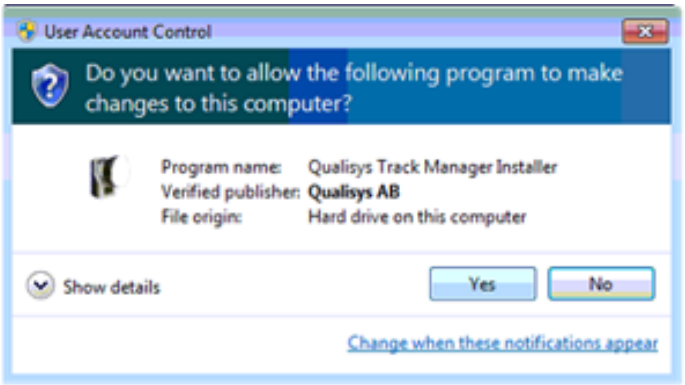


Upgrading an existing installation

If you have used the PAF module before, follow these steps to update to a new version.

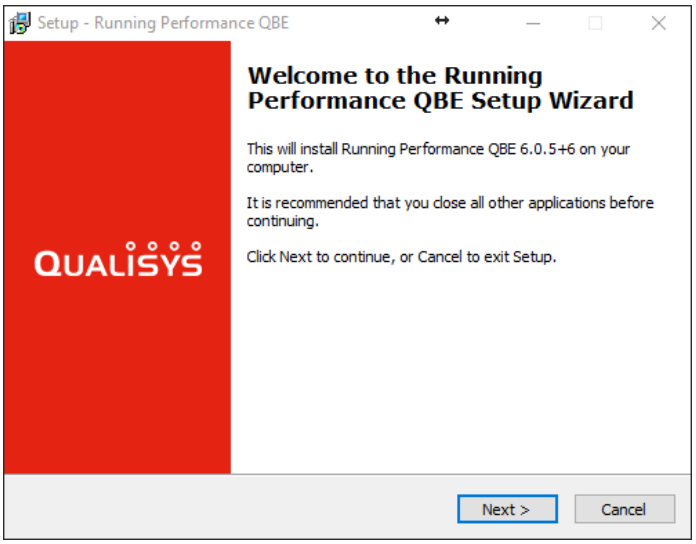
1. We recommend that you create a backup of the complete project folder. In particular, make sure to keep a backup copy of all files that you have customized since you installed the module (for example report templates or AIM models).
2. Start QTM and load any project.
3. Either compressed installer (*.qpi) or executable installer (*.exe) can be used:

a. If installing the module from qpi file, double-click the *.qpi file that you downloaded from the Qualisys client login. Allow QTM to make changes if Windows prompts you to do so:

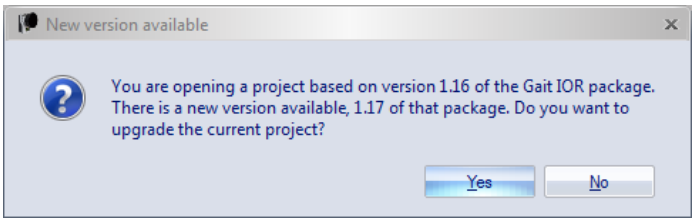


After 10-15 seconds QTM will show a “Success” message.

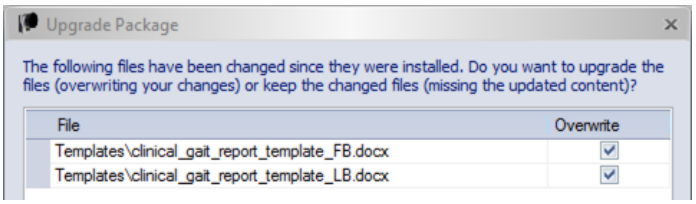
b. If installing the module from executable file, click the executable *.exe file that you downloaded from the Qualisys client login. Follow the instruction.



4. Re-start QTM and load the project that you want to update. QTM will ask if you want to upgrade the project; click Yes:



5. If you have modified template files within the PAF project and there is a conflict because the same files have also been changed in the PAF installer, the following dialog is shown:



If you check Overwrite, your changed files will be overwritten.

If you un-check Overwrite, you will keep your changed files but will miss the updated files from the module installer.

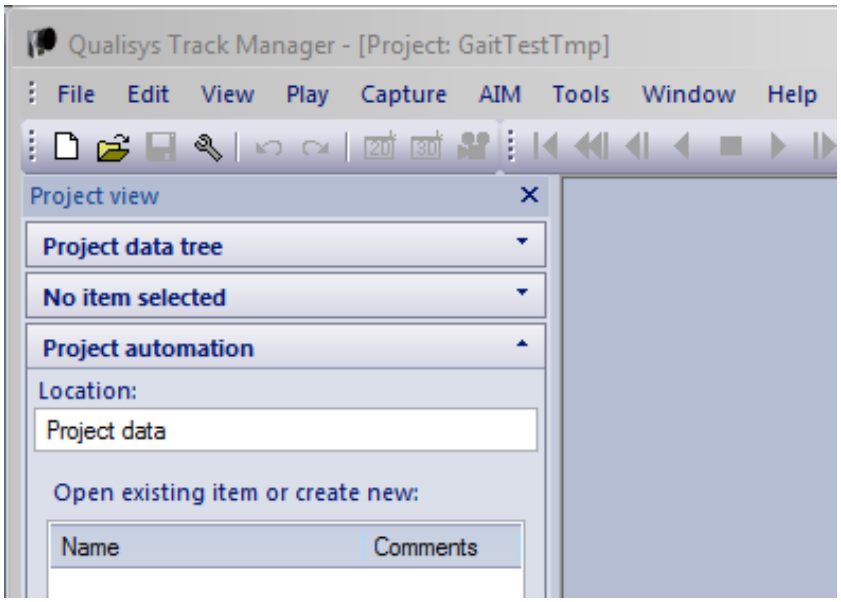
Note: for AIM files, your modified files are kept by default. If only AIM files are affected from conflicts, the dialog is not shown.

Note: if you have modified a file, but the original file has not changed between module versions, your modified file will be kept automatically.

6. The update is complete. You can verify the module version number in QTM under Help > About.

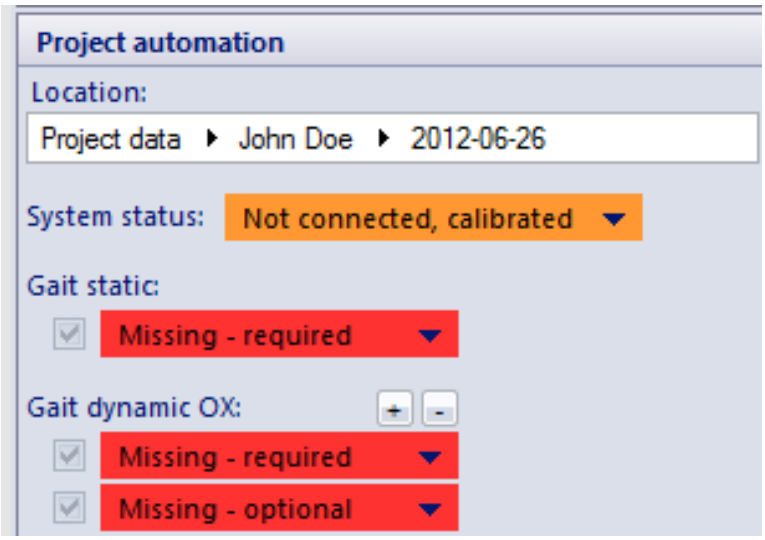
PAF Project view

The project view in a PAF project consists of three parts:

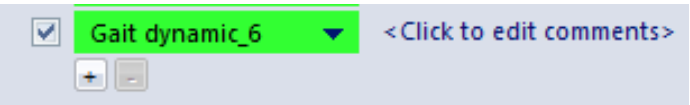


- The Project data tree shows all files that are part of the project (i.e. files contained in the Data subfolder of the project root folder).
- The Details area allows editing the properties of the selected item (for example personal data, session details, file details).
- The Project automation area shows contents depending on the selected item. If the selected item is a session, this area shows buttons that correspond to files and the Go button for data collection.
  At the top of the Project automation area there is a breadcrumb trail to navigate back to the session, subject or root level.

When a session has been selected, colored buttons appear in the Project automation area:

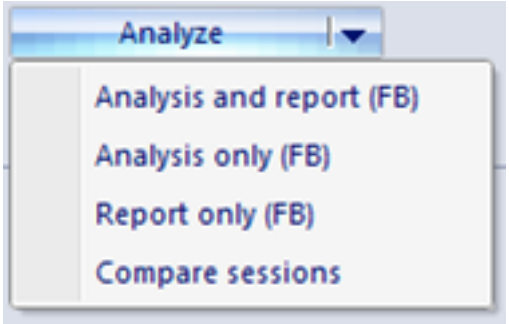


1. There are three methods to collect files:
   a. Click a red field and select Capture.
   b. Click a red field, select Edit Settings and Capture. Changes to the duration of the measurement period will be stored and be used the next time you make a measurement of the same type.
   c. Click the Go button. If you click the Go button, QTM starts at the top of the file list and records all files of the first measurement type (for example static). If you click again, it continues with all measurements of the second measurement type (for example). Activate the external trigger button in Project Options > Timing and use it to control when the captures start. At any point, click the Esc key to stop the data collection.
2. The following options are available for a recorded file:



   a. Add comments by clicking the field next to the green file name.
   b. Use the plus/minus button to show more/less file buttons (to make additional captures).
   c. Un-check a file to exclude it from the Analysis (only checked files will be exported).
3. Once the minimum number of files has been recorded, the Analyze button can be used. Click the button to run the default analysis. Depending on the PAF module, there may be multiple types of analysis. Click the triangle on

the button to show all analyses. The default analysis is topmost in the list.



Note: You can search items in the project view by using the search function (**Find...** in the Edit menu, or **Ctrl + F**).

Appendix H: 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 I: 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Oqus-和 Miqus-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。

Appendix J: Open source software compliance

Miqus

The software included in this product contains copyrighted software that is licensed under the GPL. A copy of that license is included in this document on page X. You may obtain the complete Corresponding Source code from us for a period of three years after our last shipment of this product, which will be no earlier than 2020-12-31, by sending an e-mail to support@qualisys.com or by writing to:

Qualisys AB
Kvarnbergsgatan 2
411 05 Göteborg
Sweden

This offer is valid to anyone in receipt of this information.

List of all Open Source Software components used in the Miqus camera and corresponding licenses:

Component	License	Comments/Notes
Linux kernel	GNU GPL v.2	Modified
U-boot	GNU GPL v.2	Modified
Busybox	GNU GPL v.2	Original
Dropbear	MIT style	Original
mtd-utils	GNU GPL v.2	Original
Zlib	Zlib license	Original
Lzo	GNU GPL v.2+	Original
e2fsprogs	GNU GPL v.2	Original

Licenses

GNU GENERAL PUBLIC LICENSE

Version 2, June 1991

> Copyright (C) 1989, 1991 Free Software Foundation, Inc. 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301, USA
>
> Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

Preamble

The licenses for most software are designed to take away your freedom to share and change it. By contrast, the GNU General Public License is intended to guarantee your freedom to share and change free software--to make sure the software is free for all its users. This General Public License applies to most of the Free Software Foundation's software and to any other program whose authors commit to using it. (Some other Free Software Foundation software is covered by the GNU Lesser General Public License instead.) You can apply it to your programs, too.

When we speak of free software, we are referring to freedom, not price. Our General Public Licenses are designed to make sure that you have the freedom to distribute copies of free software (and charge for this service if you wish), that you receive source code or can get it if you want it, that you can change the software or use pieces of it in new free programs; and that you know you can do these things.

To protect your rights, we need to make restrictions that forbid anyone to deny you these rights or to ask you to surrender the rights. These restrictions translate to certain responsibilities for you if you distribute copies of the software, or if you modify it.

For example, if you distribute copies of such a program, whether gratis or for a fee, you must give the recipients all the rights that you have. You must make sure that they, too, receive or can get the source code. And you must show them these terms so they know their rights.

We protect your rights with two steps: (1) copyright the software, and (2) offer you this license which gives you legal permission to copy, distribute and/or modify the software.

Also, for each author's protection and ours, we want to make certain that everyone understands that there is no warranty for this free software. If the software is modified by someone else and passed on, we want its recipients to know that what they have is not the original, so that any problems introduced by others will not reflect on the original authors' reputations.

Finally, any free program is threatened constantly by software patents. We wish to avoid the danger that redistributors of a free program will individually obtain patent licenses, in effect making the program proprietary. To prevent this, we have made it clear that any patent must be licensed for everyone's free use or not licensed at all.

The precise terms and conditions for copying, distribution and modification follow.

TERMS AND CONDITIONS FOR COPYING, DISTRIBUTION AND MODIFICATION

0. This License applies to any program or other work which contains a notice placed by the copyright holder saying it may be distributed under the terms of this General Public License. The "Program", below, refers to any such program or work, and a "work based on the Program" means either the Program or any derivative work under copyright law: that is to say, a work containing the Program or a portion of it, either verbatim or with modifications and/or translated into another language. (Hereinafter, translation is included without limitation in the term "modification".) Each licensee is addressed as "you".

Activities other than copying, distribution and modification are not covered by this License; they are outside its scope. The act of running the Program is not restricted, and the output from the Program is covered only if its contents constitute a work based on the Program (independent of having been made by running the Program). Whether that is true depends on what the Program does.

1. You may copy and distribute verbatim copies of the Program's source code as you receive it, in any medium, provided that you conspicuously and appropriately publish on each copy an appropriate copyright notice and disclaimer of warranty; keep intact all the notices that refer to this License and to the absence of any warranty; and give any other recipients of the Program a copy of this License along with the Program.

You may charge a fee for the physical act of transferring a copy, and you may at your option offer warranty protection in exchange for a fee.

2. You may modify your copy or copies of the Program or any portion of it, thus forming a work based on the Program, and copy and distribute such modifications or work under the terms of Section 1 above, provided that you also meet all of these conditions:

> **a)** You must cause the modified files to carry prominent notices stating that you changed the files and the date of any change.
>
> **b)** You must cause any work that you distribute or publish, that in whole or in part contains or is derived from the Program or any part thereof, to be licensed as a whole at no charge to all third parties under the terms of this License.
>
> **c)** If the modified program normally reads commands interactively when run, you must cause it, when started running for such interactive use in the most ordinary way, to print or display an announcement including an appropriate copyright notice and a notice that there is no warranty (or else, saying that you provide a warranty) and that users may redistribute the program under these conditions, and telling the user how to view a copy of this License. (Exception: if the Program itself is interactive but does not normally print such an announcement, your work based on the Program is not required to print an announcement.)

These requirements apply to the modified work as a whole. If identifiable sections of that work are not derived from the Program, and can be reasonably considered independent and separate works in themselves, then this License, and its terms, do not apply to those sections when you distribute them as separate works. But when you distribute the same sections as part of a whole which is a work based on the Program, the distribution of the whole must be on the terms of this License, whose permissions for other licensees extend to the entire whole, and thus to each and every part regardless of who wrote it.

Thus, it is not the intent of this section to claim rights or contest your rights to work written entirely by you; rather, the intent is to exercise the right to control the distribution of derivative or collective works based on the Program.

In addition, mere aggregation of another work not based on the Program with the Program (or with a work based on the Program) on a volume of a storage or distribution medium does not bring the other work under the scope of this License.

3. You may copy and distribute the Program (or a work based on it, under Section 2) in object code or executable form under the terms of Sections 1 and 2 above provided that you also do one of the following:

> **a)** Accompany it with the complete corresponding machine-readable source code, which must be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange; or,
>
> **b)** Accompany it with a written offer, valid for at least three years, to give any third party, for a charge no more than your cost of physically performing source distribution, a complete machine-readable copy of the corresponding source code, to be distributed under the terms of Sections 1 and 2 above on a medium customarily used for software interchange; or,

c) Accompany it with the information you received as to the offer to distribute corresponding source code. (This alternative is allowed only for noncommercial distribution and only if you received the program in object code or executable form with such an offer, in accord with Subsection b above.)

The source code for a work means the preferred form of the work for making modifications to it. For an executable work, complete source code means all the source code for all modules it contains, plus any associated interface definition files, plus the scripts used to control compilation and installation of the executable. However, as a special exception, the source code distributed need not include anything that is normally distributed (in either source or binary form) with the major components (compiler, kernel, and so on) of the operating system on which the executable runs, unless that component itself accompanies the executable.

If distribution of executable or object code is made by offering access to copy from a designated place, then offering equivalent access to copy the source code from the same place counts as distribution of the source code, even though third parties are not compelled to copy the source along with the object code.

4. You may not copy, modify, sublicense, or distribute the Program except as expressly provided under this License. Any attempt otherwise to copy, modify, sublicense or distribute the Program is void, and will automatically terminate your rights under this License. However, parties who have received copies, or rights, from you under this License will not have their licenses terminated so long as such parties remain in full compliance.

5. You are not required to accept this License, since you have not signed it. However, nothing else grants you permission to modify or distribute the Program or its derivative works. These actions are prohibited by law if you do not accept this License. Therefore, by modifying or distributing the Program (or any work based on the Program), you indicate your acceptance of this License to do so, and all its terms and conditions for copying, distributing or modifying the Program or works based on it.

6. Each time you redistribute the Program (or any work based on the Program), the recipient automatically receives a license from the original licensor to copy, distribute or modify the Program subject to these terms and conditions. You may not impose any further restrictions on the recipients' exercise of the rights granted herein. You are not responsible for enforcing compliance by third parties to this License.

7. If, as a consequence of a court judgment or allegation of patent infringement or for any other reason (not limited to patent issues), conditions are imposed on you (whether by court order, agreement or otherwise) that contradict the conditions of this License, they do not excuse you from the conditions of this License. If you cannot distribute so as to satisfy simultaneously your obligations under this License and any other pertinent obligations, then as a consequence you may not distribute the Program at all. For example, if a patent license would not permit royalty-free redistribution of the Program by all those who receive copies directly or indirectly through you, then the only way you could satisfy both it and this License would be to refrain entirely from distribution of the Program.

If any portion of this section is held invalid or unenforceable under any particular circumstance, the balance of the section is intended to apply and the section as a whole is intended to apply in other circumstances.

It is not the purpose of this section to induce you to infringe any patents or other property right claims or to contest validity of any such claims; this section has the sole purpose of protecting the integrity of the free software distribution system, which is implemented by public license practices. Many people have made generous contributions to the wide range of software distributed through that system in reliance

on consistent application of that system; it is up to the author/donor to decide if he or she is willing to distribute software through any other system and a licensee cannot impose that choice.

This section is intended to make thoroughly clear what is believed to be a consequence of the rest of this License.

8. If the distribution and/or use of the Program is restricted in certain countries either by patents or by copyrighted interfaces, the original copyright holder who places the Program under this License may add an explicit geographical distribution limitation excluding those countries, so that distribution is permitted only in or among countries not thus excluded. In such case, this License incorporates the limitation as if written in the body of this License.

9. The Free Software Foundation may publish revised and/or new versions of the General Public License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns.

Each version is given a distinguishing version number. If the Program specifies a version number of this License which applies to it and "any later version", you have the option of following the terms and conditions either of that version or of any later version published by the Free Software Foundation. If the Program does not specify a version number of this License, you may choose any version ever published by the Free Software Foundation.

10. If you wish to incorporate parts of the Program into other free programs whose distribution conditions are different, write to the author to ask for permission. For software which is copyrighted by the Free Software Foundation, write to the Free Software Foundation; we sometimes make exceptions for this. Our decision will be guided by the two goals of preserving the free status of all derivatives of our free software and of promoting the sharing and reuse of software generally.

NO WARRANTY

11. BECAUSE THE PROGRAM IS LICENSED FREE OF CHARGE, THERE IS NO WARRANTY FOR THE PROGRAM, TO THE EXTENT PERMITTED BY APPLICABLE LAW. EXCEPT WHEN OTHERWISE STATED IN WRITING THE COPYRIGHT HOLDERS AND/OR OTHER PARTIES PROVIDE THE PROGRAM "AS IS" WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE PROGRAM IS WITH YOU. SHOULD THE PROGRAM PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.

12. IN NO EVENT UNLESS REQUIRED BY APPLICABLE LAW OR AGREED TO IN WRITING WILL ANY COPYRIGHT HOLDER, OR ANY OTHER PARTY WHO MAY MODIFY AND/OR REDISTRIBUTE THE PROGRAM AS PERMITTED ABOVE, BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY GENERAL, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE PROGRAM (INCLUDING BUT NOT LIMITED TO LOSS OF DATA OR DATA BEING RENDERED INACCURATE OR LOSSES SUSTAINED BY YOU OR THIRD PARTIES OR A FAILURE OF THE PROGRAM TO OPERATE WITH ANY OTHER PROGRAMS), EVEN IF SUCH HOLDER OR OTHER PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

END OF TERMS AND CONDITIONS

How to Apply These Terms to Your New Programs

If you develop a new program, and you want it to be of the greatest possible use to the public, the best way to achieve this is to make it free software which everyone can redistribute and change under these terms.

To do so, attach the following notices to the program. It is safest to attach them to the start of each source file to most effectively convey the exclusion of warranty; and each file should have at least the "copyright" line and a pointer to where the full notice is found.

> *one line to give the program's name and an idea of what it does.*
>
> Copyright (C) *yyyyname of author*
>
> This program is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version.
>
> This program is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.
>
> You should have received a copy of the GNU General Public License along with this program; if not, write to the Free Software Foundation, Inc., 51 Franklin Street, Fifth Floor, Boston, MA 02110-1301, USA.

Also add information on how to contact you by electronic and paper mail.

If the program is interactive, make it output a short notice like this when it starts in an interactive mode:

> Gnomovision version 69, Copyright (C) *year name of author* Gnomovision comes with ABSOLUTELY NO WARRANTY; for details type 'show w'. This is free software, and you are welcome to redistribute it under certain conditions; type 'show c' for details.

The hypothetical commands 'show w' and 'show c' should show the appropriate parts of the General Public License. Of course, the commands you use may be called something other than `show w' and `show c'; they could even be mouse-clicks or menu items--whatever suits your program.

You should also get your employer (if you work as a programmer) or your school, if any, to sign a "copyright disclaimer" for the program, if necessary. Here is a sample; alter the names:

> Yoyodyne, Inc., hereby disclaims all copyright interest in the program `Gnomovision' (which makes passes at compilers) written by James Hacker.
>
> *signature of Ty Coon*, 1 April 1989 Ty Coon, President of Vice

This General Public License does not permit incorporating your program into proprietary programs. If your program is a subroutine library, you may consider it more useful to permit linking proprietary applications with the library. If this is what you want to do, use the GNU Lesser General Public License instead of this License.

MIT License

Copyright (c) 2002-2015 Matt Johnston
Portions copyright (c) 2004 Mihnea Stoenescu
All rights reserved.

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

zlib

/* zlib.h -- interface of the 'zlib' general purpose compression library version 1.2.8, April 28th, 2013

Copyright (C) 1995-2013 Jean-loup Gailly and Mark Adler

This software is provided 'as-is', without any express or implied warranty. In no event will the authors be held liable for any damages arising from the use of this software. Permission is granted to anyone to use this software for any purpose, including commercial applications, and to alter it and redistribute it freely, subject to the following restrictions:

1. The origin of this software must not be misrepresented; you must not claim that you wrote the original software. If you use this software in a product, an acknowledgment in the product documentation would be appreciated but is not required.
2. Altered source versions must be plainly marked as such, and must not be misrepresented as being the original software.
3. This notice may not be removed or altered from any source distribution.

Jean-loup Gailly	Mark Adler
jloup@gzip.org	madler@alumni.caltech.edu

*/

Revision history

Current version of QTM: 2020.2

See release notes in the QTM documentation folder (e.g. C:\Program Files (x86)\Qualisys\Qualisys Track Manager\Documentation) or at www.qualisys.com (login required).

Glossary

2D
Two dimensions

2D tracking
Tracker that uses the 2D data of a single camera to calculate trajectories in a plane.

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.

6DOF
Six degrees of freedom

6DOF body (rigid body)
Body that is defined by fixed points on the body and a local coordinate system, i.e. location and rotation.

6DOF tracking
Tracker that calculates the position and rotation of a rigid body in the 3D view.

A/D board (analog board)
Analog/Digital board, which converts an analog signal to a digital.

AIM (Automatic Identification of Markers)
Process that automatically identifies the trajectories as being part of a defined trajectory.

Analog capture
QTM can capture analog voltage data in synchronization with the motion capture data. If you have an analog board.

Analog output
With analog output 6DOF data can be used as feedback to an analog control system. If you have an analog output board.

Analog output board
Board which converts a digital value to an analog signal.

Analog synchronization cable
Cable that connects the camera system with the analog board for synchronization of the start.

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.

Camera ray
The 2D position of a marker projected into the 3D space based on the position and orientation of the camera.

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.

D/A board
Digital/analog board, which converts a digital value to an analog signal.

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 timebase
Device that controls the frame rate of the camera system.

External trigger
Device that triggers the motion capture system to start the measurement.

FBX
FBX (filmbox) is a widely used file format for exchange of 3D and skeleton data for animation software.

Field of view (FOV)
The camera'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).

Fixed camera calibration
Calibration method which is used for large systems with fixed cameras.

Focus
Changes the focal length of the camera to achieve a clear image.

Force plate
Plate that can measure the force that is applied on top of it.

Frame
Single exposure of the camera system.

Frame rate
Frequency of the motion capture.

Gap
Missing part within a trajectory.

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.

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 Diode

Linearization
Correction data which is needed for each camera to make the capture as good as possible.

Local coordinate system
Coordinate system of a 6DOF body.

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.

Marker mode
The default mode in 2D view windows, which shows the markers captured by the camera.

Markerless mocap
Video based motion tracking without the use of markers.

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.

Mesh
Wavefront 3D object (.obj) files and associated .mtl and texture files for visualization of 3D objects.

Motion capture
Measurement which records a motion.

Open GL
Standard graphic presentation language.

Pitch
Rotation around the Y-axis in Qualisys standard rotation.

Plot window
Window with data plots.

Pretrigger
Setting in QTM where the frames before the trigger are saved in the camera'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 Qualisys cameras.

Real-time output (RT output)
Function which exports 3D, 6DOF and Analog data in real-time to a remote computer.

Reference marker
Special kind of active marker which is used in fixed camera systems and is visible at long distances.

Reference structure
The L-shaped part in the calibration kit of the Wand calibration.

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 (6DOF)
The average of the errors of each measured marker compared to the 6DOF body definition.

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.

Rigid body (6DOF body)
Body that is defined by points and a local coordinate system, i.e. location and rotation.

Roll
Rotation around the X-axis in Qualisys standard rotation.

Skeleton
Series of segments organized in joint chains with hierarchical relationships.

Skeleton calibration
Function that creates a skeleton including degrees of freedom, joint boundaries, position of segments origins and position of markers expressed in their respective segment coordinate system.

Skeleton solver
Function that fits a calibrated skeleton definition to measured 3D trajectories.

Smooth
Operation applied to data in order to reduce noise while preserving important patterns.

Spike
Discontinuity between consecutive frames within a trajectory.

Subpixel
Unit used to express marker position and size in the 2D data of a Qualisys camera. The number of subpixels is obtained by multiplying the number of pixels in each dimension by a factor 64.

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, 6DOF or skeleton data.

Trajectory
3D data of a marker in a series of frames.

Trajectory editor window
Window for graphical display and editing of trajectories.

Trajectory info windows
Windows with trajectories and 3D data.

Translate
Move the center of rotation and zoom in the current 2D plane of the 3D view.

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.

Twin system
Connect two separate camera systems and merge their 3D data in one file. For example with above and under water systems.

Unidentified trajectories window
Window in QTM with unidentified trajectories.

USB
Hardware interface for connecting peripheral units to a computer.

View window
Window in QTM which shows 2D, 3D or Video views.

Virtual trajectory
Artificial trajectory added during post processing without a direct relation to measured 2D data.

Volume
The defined measurement's height, length, depth.

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.

WLAN
Wireless local area network.

Yaw

Rotation around the Z-axis in Qualisys standard rotation.

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

2

2D data
- graphic display 27
- plot 87
- TSV export 389
- TSV file format 390
- viewing data 86

2D tracking 360
- description 360
- settings 187

2D view window 27

3

3D data 63
- C3D export 397
- graphic display 48
- Matlab export 399
- Matlab file format 399
- plot 72
- TSV export 389
- TSV file format 390
- viewing data 63

3D motion capture 281
- camera placement 255
- outline 281

3D tracking 334
- description 334
- Maximum residual 185

Minimum ray count 185
Minimum trajectory length 185
parameters 184
Prediction error 184
Ray length limits 186
settings 183
test 335
troubleshooting 497
3D view window 43
bones 49
change appearance 44
contents 43
menu 57
rigid bodies 50
Skeletons 51
trajectories 48

6

6DOF analog output 346
how to use 346
settings 229
6DOF bodies (Rigid bodies) 337
creating 337
definition 338
graphic display 50
local coordinate system 341
referring coordinate system 202
settings 196
6DOF capture 281
analog output 346
real-time output 346
troubleshooting 501
6DOF data 87
6DOF versus 3D 337
calculation F - 1
Euler angles (rotation angles) 407
graphic display 50
Matlab export 399

Matlab file format 399
rotation angles 344
TSV export 389
TSV file format 392
viewing data 87
6DOF real-time output 346
description 346
settings 227
6DOF tracking 336
description 341
introduction 336
parameters 196

A

A/D board 450
channels 170
connect 450
drivers C - 2
settings 166
supported boards C - 1
synchronization C - 8
USB-1608G C - 5
USB-2533 C - 2
AIM 364
apply model 372
generate model 365
settings 193
Skeleton 354
Analog data 91
capture 450
plot 91
TSV file format 394
viewing data 91
Analyze 74
Aperture
Arqus 428
Miqus 430
Oqus 434

Tips 257
Arqus
Aperture and focus 428
Back side 415
Camera identification 256
Camera Sync Unit (back) 419
Camera Sync Unit (front) 418
Description 414
Firmware update B - 1
Front side 414
How to connect 426

B

Batch capture 283
Batch processing 329
description 329
settings 182
Blackmagic Design 464
connecting video source (Decklink Mini Recorder) 465
connecting video source (Intensity Pro) 465
installing 464
settings 466
Bones 49
create 49

C

C3D export 397
export 397
file format 398
settings 233
C3D import 102
Calibration 269
calibration kit 275
error messages 273
extended calibration 276
fixed camera calibration 279
fixed camera settings 147
outline (wand calibration) 269

perform 269
perform settings 270
reprocess 279
result 271
settings 144
transformation 149
troubleshooting 493
wand calibration 275
wand calibration tips 275
wand settings 144
Camera positioning 254
Camera rays 52
Camera Sync Unit 418
Back side 419
Front side 418
Cameras
How to set up 424
Types 409
Capture 281
6DOF real-time output 346
batch capture 283
capture mode 13
capture period 283
capture rate 123
guidelines 254
how to 281
start capture dialog 282
troubleshooting 494
Connection 253
camera connection 253
Locate system 124
troubleshooting 491
Coordinate system 43
for rigid body data 202
global (defined by calibration) 144
local 341
transformation 149
view in 3D views 43

D

D/A board 346
Data export 389
C3D export 397
C3D file format 398
C3D settings 233
description 389
FBX export 406
FBX settings 238
MAT export 399
MAT file format 399
Matlab settings 234
TSV export 389
TSV file format 390
TSV settings 231
Data info window 85
data types 86
menu 85
plot 85
Data types 86
2D data 86
3D data 63
6DOF data 87
analog data 91
force data 92
Skeleton data 89
Discarded trajectories 63
delete trajectories 72
window 63

E

EMG device 463
Cometa D - 33
Delsys D - 26
Mega D - 17
Noraxon D - 1

Euler angles (Rotation angles) 407
calculation from the rotation matrix F - 1
custom definition F - 3
description 407
example 344
Events 383
Camera Sync Unit 157
External trigger 156
Wireless trigger 155
Exposure time 128
Tips 258
External timebase 476
connect 476
how to use 476
settings 159
with bursts of signals 479
External trigger 475
how to use 475

F

FBX export 406
Settings 238
Firmware update 443
QFI B - 1
when locating system 443
when opening a new file 443
Fixed camera calibration 279
how to use 279
settings 147
Focus
Arqus 428
Miqus 430
Oqus 434
Tips 257
Force calculation 386
export 397
settings 211
troubleshooting 500

viewing force 386
Force plates 454
AMTI Gen5 460
connect 454
force plate control (Kistler) 173
settings (COP) 227
settings (force plate types) 211
settings (location) 224
settings (use) 210
supported force plates 462
Frame rate 123

G

Gap fill 377
How to 378
Kinematic 380
Linear 379
Methods 378
Polynomial 379
Relational 379
settings 193
Static 379
Virtual 379

H

Hardware 6
A/D board 450
Arqus 414
configuration 10
D/A board 346
EMG device 463
external timebase 476
external trigger 475
force plate 454
Miqus 416
Pretrigger 475
requirements 4
video 473

High-speed video (Oqus) 289
description 289
Settings (advanced) 134
Settings sidebar 31

I

Identification 364
automatic (AIM) 364
manual 362
settings (AIM) 193

L

Label list 77
Labeled trajectories window 63
Linearization 263
how to 263
settings 141

M

Marker (3D view) 43
data 48
Marker (IR) 261
maintenance 262
passive/active 261
placement 262
settings 128
sizes 262
Tips on settings 258
Marker discrimination 132
Marker threshold 129
Tips 258
Markerless mocap
How to set up the system 431
Matlab export 399
MAT export 399
MAT file format 399
Measurement range (timeline) 59
Mega EMG D - 17

Mesh
Rigid body settings 205
Static 247
Static mesh settings 248
Miqus
Aperture and Focus 430
Camera back side 417
Camera front side 416
Camera Sync Unit (back) 419
Camera Sync Unit (front) 418
Description 416
Firmware update B - 1
How to connect 428
Identification 256
Order 256
Mixed system
How to connect 435
Mobile apps 317
QTM Remote 317
Viewfinder 317

N

Noraxon EMG D - 1

O

Oqus
Aperture and Focus 434
configurations A - 4
connectors 421
display 420
firmware update B - 1
functions A - 9
high-speed 289
How to connect 432
Oqus Sync Unit 423
Oqus Sync Unit 423
How to use 486
Overlapping trajectories 66

P

PCI boards
A/D board 450
Pitch F - 1
definition F - 3
example 344
view 87
Plot 94
3D data 72
data in Data info window 85
window 94
Pretrigger 158
hardware with analog capture 475
How to use 475
Settings 158
Preview mode 13
Processing 326
AIM 364
available steps 326
batch processing 329
calculate force data 386
data export 389
settings 182
tracking 3D 334
tracking 6DOF 341
Project options 119
Project view 19
Projects 14
backup 22
creating 17
presets 22
using 14
PTP sync 480, 482
how to use 480, 482

Q

QDS 437
advanced 441
menu 437
Qualisys DHCP server message 442
wizard (network) 439
QFI B - 1
Qualisys video 286
3D data overlay 295
calibration 287
high-speed video (Oqus) 289
How to connect 431
how to use 286
In-camera MJPEG 289
Markerless mocap 431
streaming video capture 287

R

Rays 52
Real-time output 313
description 313
Github 313
how to use 315
SDKs 313
settings 227
Unity 313
Unreal 313
Recalibration 279
Reference marker 279
Reference structure 275
Retracking 327
Rigid body (6DOF body) 337
creating 337
definition 338
graphic display 50
local coordinate system 341
referring coordinate system 202

settings 196
Roll F - 1
definition F - 3
example 344
view 87
Rotation angles (Euler angles) 407
calculation from the rotation matrix F - 1
custom definition F - 3
description 407
example 344

S

Skeleton solver
Calibration 356
Data 89
Export and streaming 360
Extra markers 353
FBX export 238, 406
Graphic display 51
How to measure 360
Kinematic gap fill 380
Marker label mapping 353
Segment names 353
Settings 195
T-pose 356
TSV export format 393
View data 89
Skeleton tracking 350
AIM 354
Introduction 350
Skeletons 51
Smoothing 381
Butterworth 381
Methods 381
Moving average 381
SMPTE
How to use 488
Synchronization 488

Software masks
How to use 331
Spikes 380
Detection 381
Smoothing 381
Threshold 381
Store real-time data 11, 284
Streaming video 287
Settings (advanced) 134
Settings sidebar 31
Synchronization 154
Audio 488
Oqus Sync Unit 486
Pretrigger 475
Settings 154
SMPTE 488
Synchronization output 162
Measurement time 165
settings 162
Synchronizing external hardware 484

T

Timeline control bar 59
Timestamp 162
settings 162
Timing
external timebase 476
external trigger 475
Trace 44
display (all) 57
display (individual) 68
range 59
Tracking 334
definition (2D) 360
definition (3D) 334
definition (6DOF) 336
Definition (skeleton) 350
parameters (3D) 184

parameters (6DOF) 196
retracking 327
settings (2D) 187
settings (3D) 183
settings (6DOF) 196
test 335
troubleshooting 497
Trajectories 63
analyze data 74
delete 72
display 68
in 3D views 48
label lists 77
overlapping 66
plot data 72
select and move 65
split part 72
windows 63
Trajectory Editor window 79
Gap fill 377
Menu 84
Points of Interest sidebar 82
Settings sidebar 83
Shortcuts 118
Smoothing 380
Spikes 380
Toolbar 80
Trajectory info windows 63
data in 63
menu 68
Trigger
External 156
Keyboard 155
Software 155
Wireless 155
Trigger port(s) 156
External trigger 156
Settings 156

TSV export 389
export 389
file format (3D) 390
file format (6DOF) 392
file format (Analog) 394
settings 231
Skeleton data 393
Twin systems 301
calibration 307
files 309
settings 189
setup 301

U

Upgrade firmware
QFI B - 1
USB A/D board C - 2
USB-1608G C - 5
USB-2533 C - 2

V

Video 464
Blackmagic Design 464
compression 473
how to use 464
settings 176
view window 36
View window 27
2D 27
3D 43
video 36
Virtual trajectories 382

W

Wand calibration 275
extended calibration 276
how to use 275
settings 144

tips 275
Window layout 98
Wireless trigger 155
Settings 155

Y

Yaw F - 1
definition F - 3
example 344
view 87