STONEX

Stonex Reconstructor
3D Software
User Manual

STONEX
RECONSTRUCTOR 4.4

(February 2023)-Ver.1-Rev.1

www.stonex.it

Table of contents

Introduction 8
- Contents 9
- System Requirements 11
  - Cpu known issues 13
- Installation 33
- Licensing 40
- Disclaimer 43
- Stonex End User License and Maintenance Agreement 44

Getting Started 50
- User Interface 52
  - Menu bar 54
  - Toolbox Bar and Top Toolbar 54
  - View, Navigation and Select points tools 55
  - 3D window 57
  - Dockable windows 59
  - Project Items 60
- Import 62
- LineUp 64
  - Import Point Clouds 65
  - Registration 68
    - Preregistration 71
    - Fine registration 72
  - Georeferencing 72
    - Reference points selection 73
    - Scans georeferencing 74
    - 3D Model georeferencing 75
  - Point filtering and clustering 76
- Meshes 78
  - Manage Meshes 79
  - Manage points and shapes 80
- Measures 82
- CAD Outputs 83
- Color 85
  - Cloud coloring 86
  - Camera calibration, projectors and colorization 87
- Mining 88
- Export 89

File 90
- Project Management 92
- Import 96
  - Import Reconstructor Project 97
  - Load item 97
  - Import a list of points 98
  - Import external data 99
    - CAD formats 101
      - IFC 102
- Export 103

Export Model As ... 104
Export E57 ... 108
Export polyline ... 109
Export cross section as ... 109
Export UCS to AutoCAD ... 109
Export annotations ... 109
Export pose ... 110
Export to Autodesk ReCap ... 111
Navigation ... 111
Predefined Views ... 113
Navigation Modes ... 116
Pivot mode ... 116
Walk mode ... 118
Bubble mode ... 118
Zoom ... 119
Move to viewport ... 119
Recent camera positions ... 119
Screen Settings ... 120
Display and navigation Options ... 120
Light settings ... 123
View Parameters ... 125
Data management ... 126
LineUp ... 127
Scan Processing Wizard ... 128
Wizard Settings ... 129
Cloud-to-cloud registration ... 135
Automatic Target-free Pre-registration ... 136
Sketch Window ... 139
Manual Pre-registration ... 140
Manual Pre-registration among grid point clouds ... 140
Manual Pre-registration among models ... 142
ICP Registration ... 144
Bundle Adjustment ... 149
Bundle Adjustment Report ... 151
Scan alignment per groups ... 153
Target-based Registration ... 154
Target registration ... 155
Targets Editor ... 156
Manage Control Points ... 157
Manage Circular Targets ... 157
Manage Spherical Targets ... 159
Targets Registration Report ... 160
Georeferencing ... 162
Point clouds georeferencing ... 162
Point list registration ... 164
Compute Registration ... 165
Registration report dialog ... 166
Targets, annotations and points conversion ... 167
UCS & positioning ... 168
Manual positioning ... 169

Pose transform 170
Pose 171
Determine the cloud's vertical direction 176
Create/edit UCS 176
Create UCS from point list 179
Create UCS from this pose 180
Define project reference system 181
Selection & Editing 182
Select points 183
Point selection with polyline 186
Cloud Processing 187
Pre-processing Point Clouds 188
Merge Clouds Into One 192
Merge clouds and level3D density 192
Resample 194
Simplify Points 194
Extract Edges 194
Fill Holes 195
Hide Black points 195
Edit 2D 196
Remove duplicate points 197
Invert Normals 197
Restore Deleted Points 199
Restore Raw Data 200
Meshing 201
Mesh creation 202
Multiresolution Meshing 203
Meshing from predefined view 206
3D Meshing 208
Topographic meshing 211
Mesh point list 214
Mesh manipulation 214
Mesh Editor 215
Merge mesh into one 220
Mesh selection with 3D polyline 220
Convert mesh to point cloud 222
Get mesh border as polyline 222
Mesh selection from current view point 222
Geometric Shapes 223
Manage points 224
Edit point list 224
Export point list 225
Polyline from point list 226
Fit shape to point list 227
Basic shapes 227
Create/edit plane 227
Create/edit segment 230
Create/edit polyline 232
Second order shapes 235
Create/edit circle 235

Create/edit sphere 237
Create/edit cylinder 239
Create/edit cone 241
Shapes interactions 243
Distance between shapes 244
Intersect geometric shapes 244
Photo & Color 245
Histogram Dialog 248
Colors Mapping 249
Color with altitude 252
Inclination from plane 253
Color with range 253
Color with ambient light 254
Compute 3D reflectance 255
Add color layer from image 256
Merge or split layers 256
Color clouds with photos 257
Create projector 258
Edit (projector) image 259
Camera calibration on a model 260
Camera calibration using 3D points 263
Calibrate camera intrinsics 264
Create Panorama 266
UAV and imaging 268
Import GeoTIFF 268
Create texture map 269
Results & Analyses 270
Measures & Notes 271
Readout window 272
Annotation 274
Distance 275
Angle Tool 277
Open Angle Tool 278
Elevations & Plans 278
Create Orthophoto 279
Create X-Ray Orthophoto 282
Orthophoto Viewer 286
Create Camera 290
Place Here Survey Point 291
Virtual scan 291
Exporting orthophotos to AutoCAD 294
Cross sections 296
Cross section 301
Contours 302
Quick profile 302
Cross sections from planes 302
Section from plane 303
Tunnel cross sections 303
DEM points 304
Areas & Volumes 304

Area ... 305
Volume from plane ... 306
Volume PDF report ... 307
Cut&fill volumes ... 309
Cut and fill by altitude report ... 310
Inspection ... 311
Drawing tools ... 315
Video Record ... 316
Flythrough Editor ... 317
Make movie ... 318
Flythrough from point list ... 318
Save snapshot ... 319
Manage Units Of Measures ... 319
Special tools ... 322
Tunnel survey ... 323
Windows ... 324
Project Window ... 325
Recently used items dialog ... 328
Property Window ... 329
Log Window ... 330
Information window ... 331
GUI options ... 332
Custom toolbox ... 333
Add-ons ... 334
Reconstructor Viewer ... 335
CONSTRUCTION add-on ... 336
MINING add-on ... 336
COLOR MAPPING add-on ... 337
Help ... 337
Online Help ... 338
Offline PDF Manual ... 339
License Manager ... 339
Show Commands Dashboard ... 340
Memory Usage Report ... 341
HardwareIinfo ... 341
About Reconstructor ... 342
Third Party Libraries ... 343
LASlib ... 344
Check For Updates ... 346
Open Log Folder ... 346
Crash report ... 347
Glossary ... 347
Annotations ... 348
Cameras ... 348
Flythrough ... 350
Groups ... 352
Planes ... 353
Point Clouds ... 354
Polylines ... 356
Projectors ... 357

Triangle Meshes 357
User Coordinate System (UCS) 360

Reconstructor® 4.4 Manual

Last edit: 2023/01/24

Reconstructor® is a software to combine 3D information from different sensors, prepare intermediate representations and extract information. The main advantage is the independency from a specific sensor and the comprehensive amount of tools and features that enable easy extraction of quality results in many fields: mining, surveying, construction, architecture, cultural heritage, forensics, BIM, tunneling, etc.

1. Reconstructor® imports all the most used laser scanner formats in the market. The workflow begins with the LineUp® wizard that easily allows you to import, pre-process, and automatically register any amount of data with state-of-the-art speed and precision.
2. After that, your data are processed enough to easily extract results: basic measures and notes, elevations and plans, cross sections, areas and volumes, geometrical check and change detection, tunnel analyses, flythrough videos, and so on.
3. Furthermore, Reconstructor® has all the features needed to deliver more elaborated results: tools for models positioning, editing of point clouds, fitting of geometric primitives, model filtering and clustering, meshing, mesh editing, coloring with calibrated photos, UAV images processing, etc.

Please click on one of the following sub-topics to proceed:

- Contents
- System Requirements
- Installation
- Licensing
- Disclaimer
- Stonex End User License and Maintenance Agreement
- Getting Started

The following conventions have been used in this manual:

- Main processes and parameters will be bold
- Commands will appear in italic, during explanations
- Clickable links will be blue colored. Use these links to consult other items in the manual or external links.
- This icon indicates an important tip for you
- Pay attention to this information/suggestion

Reconstructor® is developed by GEXCEL

Contents

The present on-line help is organized as follows:

i. An introduction about the installation and the applying license procedures of the software

ii. A Getting Started section to easily learn how to work with Reconstructor® and the main features of its user interface.

iii. An explanation of the functionalities of Reconstructor®, according to the voices in the menu bar. These are therefore the main chapters of this section:

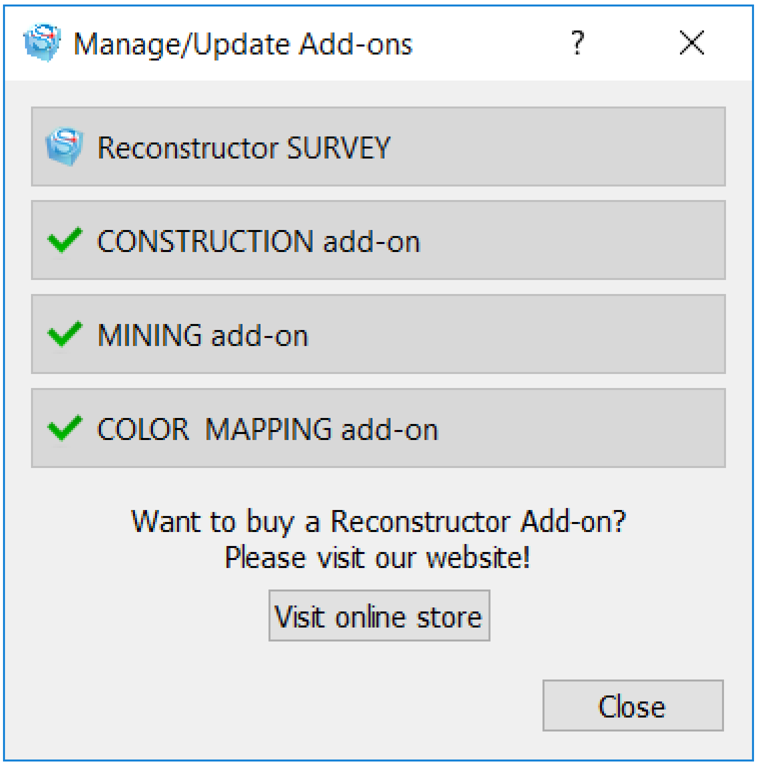
- File
- Navigation
- Data management
- Results & analyses
- Special tools
- Windows
- Add-ons
- Help

Note: some commands are not available in all software configuration, pay attention to your license!
The Reconstructor® Viewer has only a few commands active, all of them are included in Reconstructor®.

You can find all the Reconstructor® commands according to your active license in the *Command Dashboard* (Ctrl+d).

See *Help* menu -> *License Manager*

In *Add-ons* menu you can chose Add-ons functionalities according to your license.



Web references

Reconstructor® page in Stonex website.

Reconstructor® System requirements

Minimal Requirements

- Processor: 64 bit x86_64 CPU
- Main memory (RAM): 8 GB
- Graphics card that supports OpenGL 3.3 with an up to date driver, still within its vendor's support lifecycle
- Windows 10 or Windows 11
- Mouse with two buttons plus a clickable scroll wheel
- Screen resolution: 1440x960 or higher (see "Display scaling" below)

On computers with the minimal requirements, Reconstructor® will be able to work on projects with limited amount of point clouds or meshes.
Compatibility with hardware, drivers or operating system software that is no longer actively supported by its vendor cannot be guaranteed.

Recommended Requirements

- AMD Ryzen 7/Ryzen 9 or Intel Core i7/ Core i9/Xeon or equivalent, 4 or more physical cores, on a single CPU socket (see "Multi-socket processors support" below)
- Main memory (RAM): 32 GB
- Graphics card that supports OpenGL 4.0 or higher, 6 GB dedicated GPU memory, single GPU, with an up to date driver, still within its vendor's support lifecycle [1]
- Windows 10 or Windows 11
- Mouse with two buttons plus a clickable scroll wheel
- Screen resolution: 1920x1080 or higher (see "Display scaling" below)
- Hard disk: SSD or RAID10 HDD array, for optimal data read/write performance

MULTI-SOCKET PROCESSORS SUPPORT

Generally speaking, the more physical cores your processor has, the faster Reconstructor® runs. However, this only applies to cores physically located on a single processor.
Systems with multiple processor socket, have dedicated RAM memory slots for each processor socket or group of sockets (this is called a node). Access from a processor to its dedicated RAM is fast, while access to the RAM dedicated to another node is slower. This type of architecture is called NUMA (Non Uniform Memory Access).

Compatibility check

To verify how many NUMA nodes your system has, download the free Reconstructor Viewer, open it, and select Help -> Hardware Info from the top menu. A dialog pops up with information about your graphics card and your processor. Make sure that the value "Number of NUMA nodes" is 1.
The use of Reconstructor on systems with more than 1 NUMA node causes a notable slowdown with most operations that rely on fast access to large amounts of data (importing scans, pre-processing, etc).

Workaround for unsupported multi-socket processors

When running on systems where the NUMA architecture cannot be disabled from the system BIOS, it is possible to configure Reconstructor's thread affinity from the Windows Task Manager to run on the cores of a single processor.
Using less cores with fast memory access is preferable to using more cores with slower memory access.

Display scaling

Some displays have a high pixel density and require scaling for optimal visualization.
In such cases, the amount of usable logical screen space is calculated by dividing the physical display resolution by the scaling factor.
For example, a 1920x1080 FHD 15.6" display with a 125% scaling factor has a logical resolution of 1920 / 1.25 x 1080 / 1.25 = 1536x864 pixels.

[1] *GRAPHIC CARD SETTING FOR RECONSTRUCTOR*

If your PC has more than one graphic card (i.e. integrated and dedicated graphic card) combine the software with the better graphic card by using its control panel. Remember to update the graphic card drivers before to installing Reconstructor®.

See Cpu known issues chapter for details on CPU compatibility.

Reconstructor - CPU known issues

Systems with multiple sockets:

- NUMA is enforced by the hardware (each CPU socket is physically connected to a separate memory controller, and inter-CPU memory access is possible but highly penalized.
- Not much can be done, other than set Rec affinity to a single NUMA node, and only use the cores that belong to said node (minimal changes to our multithreading core count detection are required).
- Using multiple CPUs (multiple NUMA nodes) efficiently would require changes to the implementation of our algorithms, to be aware of that.

Systems with only one socket:

- Certain somewhat recent Xeon CPUs allow the creation of virtual NUMA nodes (clusters of cores) within the same die.
- In some cases this is purely virtual, and can be disabled at the BIOS level. In some other cases this is tied to the physical architecture of the die, and while it may be possible to disable it at the BIOS level, it might or might not yield benefits to Rec (this would need to be tested with one such computer, benchmarked under different configurations)

Known issues

Processors by family

Cannon Lake (2018)

Mobile

Family	Cores	Models	Known issues
Core i3	2/4	i3-8121U	Low-end[1][2]

Whiskey Lake (2018)

Mobile

Family	Cores	Models	Known issues
Core i7	4/8	i7-8565U	
Core i5	4/8	i5-8265U	Low-end[1]
Core i3	2/4	i5-8145U	Low-end[1][2]

Coffee Lake (2017-2018)

Desktop

Family	Cores	Models	Known issues
Core i7	6/12	i7-8086K, i7-8700K, i7-8700, i7-8700T	
Core i5	6	i5-8600K, i5-8600, i5-8500, i5-8400, i5-8600T, i5-8500T, i5-8400T	Low-end[1]
Core i3	4	i3-8350K, i3-8300, i3-8100, i3-8300T, i3-8100T	Low-end[1]
Pentium Gold	2/4	G56##, G55##, G54##, G55##T, G54##T	Low-end[1][2]
Celeron	2	G49##, G49##T	Low-end[1][2]

Workstation

Family	Cores	Models	Known issues
Xeon E	6/12	E-2186G, E-2176G, E-2136	
Xeon E	6	E-2126G	
Xeon E	4/8	E-2174G, E-2144G, E-2134	
Xeon E	4	E-2124G, E-2124	

Mobile

Family	Cores	Models	Known issues
Xeon E	6/12	E-2186M, E-2167M	
Core i9	6/12	i9-8950HK	
Core i7	6/12	i7-8850H, i7-8750H, i7-8700B	
Core i7	4/8	i7-8559U	
Core i5	6	i5-8500B, i5-8400B	Low-end[1]
Core i5	4/8	i5-8400H, i5-8300H, i5-8269U, i5-8259U	Low-end[1]
Core i3	4	i3-8100H	Low-end[1]
Core i3	2/4	i3-8109U	Low-end[1][2]

Desktop

Family	Cores	Models	Known issues
Core i9	8/16	i9-9900K	
Core i7	8	i7-9700K	
Core i5	6	i5-9600K	Low-end[1]

Kaby Lake (2016-2017)

Desktop

Family	Cores	Models	Known issues
Core i7	4/8	i7-77##X, i7-77##K, i7-77##, i7-77##T	
Core i5	4	i5-76##X, i5-76##K, i5-76##, i5-75##, i5-74##, i5-76##T, i5-75##T, i5-74##T	Low-end[1]
Core i3	2/4	i3-73##K, i3-73##, i3-71##, i3-73##T, i3-71##T, i3-71##TE, i3-71##E	Low-end[1][2]
Pentium	2/4	G46##, G45##, G46##T, G45##T	Low-end[1][2]
Celeron	2	G39##, G39##T	Low-end[1][2]

Mobile

Family	Cores	Models	Known issues
Xeon E3	4/8	E3-1535M, E3-1505M	
Core i7	4/8	i7-8809G, i7-8709G, i7-79##HQ, i7-78##HQ, i7-78##HK, i7-77##HQ, i7-8650U, i7-55##U	
Core i7	2/4	i7-76##U, i7-75##U, 7Y7#, i7-8500Y	Low-end[2]
Core i5	4/8	i5-8305G, i5-8350U, i5-8250U	Low-end[1]
Core i5	4	i5-74##HQ, i5-73##HQ	Low-end[1]
Core i5	2/4	i5-73##U, i5-72##U, 7Y5#, i5-8210Y, i5-8200Y	Low-end[1][2]

Core i3	2/4	i3-71##H, i3-8130U, i3-71##U	Low-end[1][2]
Core m3	2/4	7Y3#, 8100Y	Low-end[1][2]
Pentium	2/4	44##U, 44##Y	Low-end[1][2]
Celeron	2	39##U, 38##U	Low-end[1][2]

Server

Family	Cores	Models	Known issues
Xeon E3	4/8	E3-1285v6, E3-1280v6, E3-1275v6, E3-1270v6, E3-1245v6, E3-1240v6, E3-1230v6	
Xeon E3	4	E3-1225v6, E3-1220v6	
Xeon E3 (embedded)	4/8	E3-1505Lv6	

Skylake (2015)

Desktop

Family	Cores	Models	Known issues
Core i9	18/36	i9-7980XE, i9-9980XE	NUMA[4]; CoD[5]; Memory controller[6]
Core i9	16/32	i9-7960X, i9-9960X	NUMA[4]; CoD[5]; Memory controller[6]
Core i9	14/28	i9-7940X, i9-9940X	NUMA[4]; CoD[5]; Memory controller[6]
Core i9	12/24	i9-7920X, i9-9920X	NUMA[4]; CoD[5]
Core i9	10/20	i9-7900X, i9-9900X, i9-9820X	NUMA[4]; CoD[5]
Core i7	8/16	i7-7820X, i7-9800X	

Core i7	6/12	i7-7800X	
Core i7	4/8	i7-67##K, i7-67##R, i7-67##, i7-67##T	
Core i5	4	i5-66##K, i5-66##R, i5-65##R, i5-66##, i5-65##, i5-64##, i5-66##T, i5-65##T, i5-64##T, i5-64##P	Low-end[1]
Core i3	2/4	i3-63##, i3-61##, i3-63##T, i3-61##T, i3-60##P	Low-end[1][2]
Pentium	2	G45##, G44##, G45##T, G44##T, G44##TE	Low-end[1][2]
Celeron	2	G39##, G39##TE, G39##T	Low-end[1][2]

Mobile

Family	Cores	Models	Known issues
Xeon E3	4/8	E3-1575Mv5, E3-1545Mv5, E3-1535Mv5, E3-1505Mv5	
Core i7	4/8	i7-69##HQ, i7-68##HQ, i7-68##HK, i7-67##HQ	
Core i7	2/4	i7-66##U, i7-65##U	Low-end[2]
Core i5	4	i5-64##HQ, i6-63##HQ	Low-end[1]
Core i5	2/4	i5-63##U, i5-62##U	Low-end[1][2]
Core i3	2/4	i3-61##U, i3-60##U, i3-61##H	Low-end[1][2]
Core m7	2/4	6Y7#	Low-end[1][2]
Core m5	2/4	6Y5#	Low-end[1][2]
Core m3	2/4	6Y3#	Low-end[1][2]
Pentium	2/4	44##U, 44##Y	Low-end[1][2]

Celeron	2	G39##E, 39##U	Low-end[1][2]

Workstation

Family	Cores	Models	Known issues
Xeon W	18/36	W-2195	NUMA[4]; CoD[5]; Memory controller[6]
Xeon W	14/28	W-2175	NUMA[4]; CoD[5]; Memory controller[6]
Xeon W	10/20	W-2150B, W-2155	NUMA[4]; CoD[5]
Xeon W	8/16	W-2140B, W-2145	
Xeon W	6/12	W-2133, W-2135	
Xeon W	4/8	W-2123, W-2125	
Xeon W	4	W-2102, W-2104	

Server

Family	Cores	Models	Known issues
Xeon Platinum	28/56	8180, 8180M, 8176, 8176F, 8176M, 8173M	Multi-CPU[3]; NUMA[4]; Memory Controller[6]
Xeon Platinum	26/52	8170, 8170M, 8164, 8167M	Multi-CPU[3]; NUMA[4]; Memory Controller[6]
Xeon Platinum	24/48	8168, 8163, 8160, 8160F, 8160M, 8160T	Multi-CPU[3]; NUMA[4]; Memory Controller[6]
Xeon Gold	22/44	6161, 6152	Multi-CPU[3]; NUMA[4]; Memory Controller[6]
Xeon Gold	20/40	6148, 6148F, 6145, 6138, 6138T, 6138F	Multi-CPU[3]; NUMA[4]; Memory Controller[6]
Xeon Gold	18/36	6154, 6150, 6140, 6140M	Multi-CPU[3]; NUMA[4];

			Memory Controller[6]
Xeon Platinum	16/32	8153	Multi-CPU[3]; NUMA[4]; Memory Controller[6]
Xeon Gold	16/32	6142, 6142F, 6142M, 6130, 6130F, 6130T	Multi-CPU[3]; NUMA[4]; Memory Controller[6]
Xeon Gold	14/28	6132	Multi-CPU[3]; NUMA[4]; Memory Controller[6]
Xeon Gold	14/28	5120, 5120T, 5119T, 5117, 5117F	Multi-CPU[3]; NUMA[4]; Memory Controller[6]
Xeon Platinum	12/24	8158	Multi-CPU[3]; NUMA[4]
Xeon Gold	12/24	6146, 6136, 6126, 6126F, 6126T	Multi-CPU[3]; NUMA[4]
Xeon Gold	12/24	5118	Multi-CPU[3]; NUMA[4]
Xeon Silver	12/24	4116, 4116T	Multi-CPU[3]; NUMA[4]
Xeon Gold	10/20	5115	Multi-CPU[3]; NUMA[4]
Xeon Silver	10/20	4114, 4114T	Multi-CPU[3]; NUMA[4]
Xeon Gold	8/16	6144, 6134, 6134M	Multi-CPU[3]; NUMA[4]
Xeon Silver	8/16	4110, 4109T, 4108	Multi-CPU[3]; NUMA[4]
Xeon Gold	6/12	6128	Multi-CPU[3]; NUMA[4]
Xeon Bronze	8	3106	Multi-CPU[3]; NUMA[4]
Xeon Platinum	4/8	8156	Multi-CPU[3]; NUMA[4]
Xeon Gold	4/8	5122	Multi-CPU[3]; NUMA[4]
Xeon Silver	4/8	4112	Multi-CPU[3]; NUMA[4]
Xeon Bronze	6	3104	Multi-CPU[3]; NUMA[4]
Xeon E3	4/8	E3-1280v5, E3-1275v5, E3-1270v5, E3-1260v5, E3-1245v5,	

		E3-1240v5, E3-1240Lv5, E3-1230v5	
Xeon E3 (embedded)	4/8	E3-1505Lv5	
Xeon E3	4	E3-1235Lv5, E3-1225v5, E3-1220v5	

Broadwell (2014-2015)

Desktop

Family	Cores	Models	Known issues
Core i7	10	i7-69##X	NUMA[4]; CoD[5]
Core i7	8	i7-69##K	
Core i7	6	i7-68##K	
Core i7	4/8	i7-57##C	
Core i5	4	i5-56##C	Low-end[1]

Embedded

Family	Cores	Models	Known issues
Core i7	4/8	i7-57##R	
Core i5	4	i5-56##R, i5-55##R	Low-end[1]

Mobile

Family	Cores	Models	Known issues
Core i7	4/8	i7-59##HQ, i7-58##HQ, i7-	

		57##HQ	
Core i7	2/4	i7-56##U, i7-55##U	Low-end[2]
Core i5	2/4	i5-53##H, i5-53##U, i5-52##U	Low-end[1][2]
Core i3	2/4	i3-51##U, i3-50##U	Low-end[1][2]
Pentium	2/4	382#U	Low-end[1][2]
Core M	2/4	5Y##	Low-end[1][2]
Pentium	2	380#U	Low-end[1][2]
Celeron	2	37##U, 32##U	Low-end[1][2]

Server

Family	Cores	Models	Known issues
Xeon E5	22/44	E5-2699v4	Multi-CPU[3]; NUMA[4]; CoD[5]; Memory controller[6]
Xeon E5	20/40	E5-2698v4	Multi-CPU[3]; NUMA[4]; CoD[5]; Memory controller[6]
Xeon E5	18/36	E5-2697v4, E5-2695v4	Multi-CPU[3]; NUMA[4]; CoD[5]; Memory controller[6]
Xeon E5	16/32	E5-2697Av4, E5-2683v4	Multi-CPU[3]; NUMA[4]; CoD[5]; Memory controller[6]
Xeon E5	14/28	E5-2690v4, E5-2680v4, E5-2660v4, E5-2658v4, E5-2650Lv4, E5-2648Lv4	Multi-CPU[3]; NUMA[4]; CoD[5]; Memory controller[6]
Xeon E5	12/24	E5-2687Wv4, E5-2650v4, E5-2628Lv4	Multi-CPU[3]; NUMA[4]; CoD[5]
Xeon E5	10/20	E5-2689v4, E5-2640v4, E5-2630v4, E5-2630Lv4, E5-	Multi-CPU[3]; NUMA[4]; CoD[5]

		2618Lv4	
Xeon E5	8/16	E5-2667v4, E5-2620v4, E5-2608Lv4	Multi-CPU[3]; NUMA[4]
Xeon E5	8	E5-2609v4	Multi-CPU[3]; NUMA[4]
Xeon E5	6/12	E5-2643v4	Multi-CPU[3]; NUMA[4]
Xeon E5	6	E5-2603v4	Multi-CPU[3]; NUMA[4]
Xeon E5	4/8	E5-2637v4, E5-2623v4	Multi-CPU[3]; NUMA[4]
Xeon E5	8/16	E5-1680v4, E5-1660v4	NUMA[4]; CoD[5]
Xeon E5	6/12	E5-1650v4	NUMA[4]; CoD[5]
Xeon E5	4/8	E5-1630v4, E5-1620v4	NUMA[4]; CoD[5]
Xeon E3	4/8	E3-1285v4, E3-1285Lv4, E3-1265Lv4	
Xeon E3	4/8	E3-1284Lv4, E3-1278Lv4, E3-1258Lv4	
Xeon D	16/32	D-1587, D-1577, D-1571	Low-end[1]; NUMA[4]; CoD[5]; Memory Controller[6]
Xeon D	12/24	D-1567, D-1559, D-1557	Low-end[1]; NUMA[4]; CoD[5]
Xeon D	8/16	D-1553N, D-1548, D-1543N, D-1541, D-1540, D-1539, D-1537	Low-end[1]
Xeon D	6/12	D-1533N, D-1531, D-1528	Low-end[1]
Xeon D	4/8	D-1529, D-1527, D-1523N, D-1521, D-1520, D-1518, D-1513N	Low-end[1]
Pentium D	4/8	D1519, D1517	Low-end[1]
Pentium D	2/4	D1508	Low-end[1][2]
Pentium D	2	D1509, D1507	Low-end[1][2]

Haswell (2013)

Desktop

Family	Cores	Models	Known issues
Core i7 Extreme	8/16	i7-59##X	
Core i7	6/12	i7-59##K, i7-58##K	
Core i7	4/8	i7-47##, i7-47##K, i7-47##S, i7-47##T, i7-47##TE, i7-47##R	
Core i5	4	i5-46##, i5-45##, i5-44##, i5-46##K, i5-46##S, i5-45##S, i5-44##S, i5-46##T, i5-459#T, i5-44##T, i5-46##R, i5-45##R	Low-end[1]
Core i5	2/4	i5-457#T, i5-457#TE	Low-end[1][2]
Core i3	2/4	i3-43##, i3-41##, i3-43##T, i3-43##TE, i3-41##T	Low-end[1][2]
Pentium	2	G34##, G32##, G34##T, G33##TE, G32##T	Low-end[1][2]
Celeron	2	G18##, G18##T, G18##TE	Low-end[1][2]

Server

Family	Cores	Models	Known issues
Xeon E7	18/36	E7-8890v3, E7-8880v3, E7-8880Lv3, E7-8870v3	Multi-CPU[3]; NUMA[4]; Memory controller[6]
Xeon E7	16/32	E7-8867v3, E7-8860v3	Multi-CPU[3]; NUMA[4]; Memory controller[6]
Xeon E7	10/20	E7-8891v3	Multi-CPU[3]; NUMA[4]
Xeon E7	4/8	E7-8893v3	Multi-CPU[3]; NUMA[4]

Xeon E7	14/28	E7-4850v3	Multi-CPU[3]; NUMA[4]; Memory controller[6]
Xeon E7	12/24	E7-4830v3	Multi-CPU[3]; NUMA[4]
Xeon E7	10/20	E7-4820v3	Multi-CPU[3]; NUMA[4]
Xeon E7	8/16	E7-4809v3	Multi-CPU[3]; NUMA[4]
Xeon E5	18/36	E5-2699v3	Multi-CPU[3]; NUMA[4]; CoD[5]; Memory controller[6]
Xeon E5	16/32	E5-2698v3, E5-2698Av3	Multi-CPU[3]; NUMA[4]; CoD[5]; Memory controller[6]
Xeon E5	14/28	E5-2697v3, E5-2695v3, E5-2683v3	Multi-CPU[3]; NUMA[4]; CoD[5]; Memory controller[6]
Xeon E5	12/24	E5-2690v3, E5-2680v3, E5-2673v3, E5-2670v3, E5-2650Lv3, E5-2658v3, E5-2648Lv3	Multi-CPU[3]; NUMA[4]; CoD[5]
Xeon E5	10/20	E5-2660v3, E5-2650v3, E5-2628Lv3	Multi-CPU[3]; NUMA[4]; CoD[5]
Xeon E5	8/16	E5-2667v3, E5-2640v3, E5-2630v3, E5-2630Lv3, E5-2618Lv3	Multi-CPU[3]; NUMA[4]; CoD[5]
Xeon E5	6/12	E5-2643v3, E5-2620v3, E5-2608Lv3	Multi-CPU[3]; NUMA[4]; CoD[5]
Xeon E5	6	E5-2609v3, E5-2603v3	Multi-CPU[3]; NUMA[4]; CoD[5]
Xeon E5	4/8	E5-2637v3, E5-2623v3	Multi-CPU[3]; NUMA[4]; CoD[5]
Xeon E3	4/8	E3-1286v3, E3-1286Lv3, E3-1285v3, E3-1285Lv3, E3-1284Lv3, E3-1281v3, E3-1280v3, E3-1276v3, E3-1275v3, E3-1275Lv3, E3-1271v3, E3-1270v3, E3-1268Lv3, E3-1265Lv3, E3-1246v3, E3-1245v3, E3-1241v3, E3-1240v3, E3-1240Lv3, E3-	

		1231v3, E3-1230v3, E3-1230Lv3	
Xeon E3	4	E3-1226v3, E3-1225v3, E3-1220v3	
Xeon E3	2/4	E3-1220Lv3	Low-end[2]

Workstation

Family	Cores	Models	Known issues
Xeon E5	10/20	E5-2687Wv3	Multi-CPU[3]; NUMA[4]; CoD[5]
Xeon E5	8/16	E5-1680v3, E5-1660v3	NUMA[4]; CoD[5]
Xeon E5	6/12	E5-1650v3	NUMA[4]; CoD[5]
Xeon E5	4/8	E5-1630v3, E5-1620v3	NUMA[4]; CoD[5]
Xeon E5	4	E5-1607v3, E5-1603v3	NUMA[4]; CoD[5]

Mobile

Family	Cores	Models	Known issues
Core i7	4/8	i7-49##MX, i7-49##HQ, i7-48##HQ, i7-47##HQ, i7-49##MQ, i7-48##MQ, i7-47##MQ, i7-48##EQ, i7-47##EQ, i7-47##EC	
Core i7	2/4	i7-46##U, i7-45##U, i7-46##Y, i7-46##M	Low-end[2]
Core i5	2/4	i5-44##EC, i5-44##E, i5-43##U, i5-42##U, i5-43##M, i5-42##M, i5-43##Y, i5-42##Y, i5-42##H	Low-end[1][2]
Core i3	2/4	i3-41##U, i3-40##U, i3-41##E, i3-41##M, i3-40##M, i3-40##Y	Low-end[1][2]

Pentium	2	35##Y, 35##U, 35##M	Low-end[1][2]
Celeron	2	29##U, 29##M, 29##Y	Low-end[1][2]

Ivy Bridge (2012)

Desktop

Family	Cores	Models	Known issues
Core i7 Extreme	6/12	i7-4960X	
Core i7	6/12	i7-4930K	
Core i7	4/8	i7-48##K, i7-37##K, i7-37##, i7-37##S, i7-37##T	
Core i5	4/8	i5-35##K, i5-35##, i5-35##S, i5-35##T, i5-347#S, i5-347#	Low-end[1]
Core i5	4	i5-345#, i5-345#S, i5-33##P, i5-33##, i5-33##S	Low-end[1]
Core i5	2/4	i5-347#T	Low-end[1][2]
Core i3	2/4	i3-32##, i3-32##T	Low-end[1][2]
Pentium	2	G21##, G21##T, G20##, G20##T	Low-end[1][2]
Celeron	2	G16##, G16##T	Low-end[1][2]

Server

Family	Cores	Models	Known issues
Xeon E7	15/30	E7-8895v2, E7-8890v2, E7-8880Lv2, E7-8880v2, E7-8870v2	Multi-CPU[3]; NUMA[4]
Xeon E7	15/30	E7-4890v2, E7-4880v2, E7-4870v2	Multi-CPU[3]; NUMA[4]

Xeon E7	15/30	E7-2890v2, E7-2880v2, E7-2870v2	Multi-CPU[3]; NUMA[4]
Xeon E7	12/24	E7-4860v2, E7-4850v2	Multi-CPU[3]; NUMA[4]
Xeon E7	12/24	E7-8850v2	Multi-CPU[3]; NUMA[4]
Xeon E7	12/24	E7-2850v2	Multi-CPU[3]; NUMA[4]
Xeon E7	12	E7-8857v2	Multi-CPU[3]; NUMA[4]
Xeon E7	10/20	E7-8891v2	Multi-CPU[3]; NUMA[4]
Xeon E7	10/20	E7-4830v2	Multi-CPU[3]; NUMA[4]
Xeon E7	8/16	E7-4820v2	Multi-CPU[3]; NUMA[4]
Xeon E7	6/12	E7-8893v2	Multi-CPU[3]; NUMA[4]
Xeon E7	6/12	E7-4809v2	Multi-CPU[3]; NUMA[4]
Xeon E5	12/24	E5-4657Lv2	Multi-CPU[3]; NUMA[4]
Xeon E5	12/24	E5-2697v2, E5-2696v2, E5-2695v2, E5-2692v2, E5-2651v2	Multi-CPU[3]; NUMA[4]
Xeon E5	10/20	E5-4650v2, E5-4640v2, E5-4624Lv2	Multi-CPU[3]; NUMA[4]
Xeon E5	10/20	E5-2690v2, E5-2680v2, E5-2670v2, E5-2660v2, E5-2658v2, E5-2650Lv2, E5-2648Lv2	Multi-CPU[3]; NUMA[4]
Xeon E5	10/20	E5-2470v2, E5-2448Lv2, E5-2450Lv2	Multi-CPU[3]; NUMA[4]
Xeon E5	8/16	E5-4620v2, E5-4610v2	Multi-CPU[3]; NUMA[4]
Xeon E5	8/16	E5-2687Wv2, E5-2667v2, E5-2650v2, E5-2640v2, E5-2628Lv2	Multi-CPU[3]; NUMA[4]
Xeon E5	8/16	E5-2450v2, E5-2440v2, E5-2428v2	Multi-CPU[3]; NUMA[4]
Xeon E5	8/16	E5-1680v2	
Xeon E5	8	E5-4627v2	Multi-CPU[3]; NUMA[4]
Xeon E5	6/12	E5-4607v2	Multi-CPU[3]; NUMA[4]

Xeon E5	6/12	E5-2643v2, E5-2630v2, E5-2630Lv2, E5-2620v2, E5-2618Lv2	Multi-CPU[3]; NUMA[4]
Xeon E5	6/12	E5-2430v2, E5-2420v2, E5-2430Lv2, E5-2418Lv2	Multi-CPU[3]; NUMA[4]
Xeon E5	6/12	E5-1660v2, E5-1650v2	
Xeon E5	6/12	E5-1428Lv2	
Xeon E5	4/8	E5-4603v2	Multi-CPU[3]; NUMA[4]
Xeon E5	4/8	E5-2637v2	Multi-CPU[3]; NUMA[4]
Xeon E5	4/8	E5-1620v2	
Xeon E5	4/8	E5-1410v2	
Xeon E5	4	E5-2609v2, E5-2603v2	Multi-CPU[3]; NUMA[4]
Xeon E5	4	E5-2407v2, E5-2403v2	Multi-CPU[3]; NUMA[4]
Xeon E5	4	E5-1607v2	
Xeon E3	4/8	E3-1290v2, E3-1280v2, E3-1275v2, E3-1270v2, E3-1265Lv2, E3-1245v2, E3-1240v2, E3-1230v2	
Xeon E3	4/8	E3-1135Cv2, E3-1125Cv2, E3-1105Cv2	
Xeon E3	4	E3-1225v2, E3-1220v2	
Xeon E3	2/4	E3-1220Lv2	Low-end[2]
Pentium	2	1403v2, 1405v2	Low-end[1][2]

Mobile processors

Family	Cores	Models	Known issues
Core i7	4/8	i7-39##XM, i7-38##QM, i7-37##QM, i7-36##QM	

Core i7	2/4	i7-3689Y, i7-36##U, i7-35##U, i7-35##LE, i7-35##M	Low-end[2]
Core i5	2/4	i5-36##ME, i5-34##Y, i5-34##U, i5-33##M, i5-33##Y, i5-33##U, i5-32##M	Low-end[1][2]
Core i3	2/4	i3-32##Y, i3-32##U, i3-32##UE, i3-31##M, i3-31##ME, i3-31##C	Low-end[1][2]
Pentium	2/4	B925C	Low-end[1][2]
Pentium	2	A1018, 20##M, 21##U, 21##Y	Low-end[1][2]
Celeron	2	10##Y, 10##E, 10##M, 10##U, 10##UE, 9##UE	Low-end[1][2]

Sandy Bridge (2011)

Desktop

Family	Cores	Models	Known issues
Core i7 Extreme	6/12	i7-3970X	
Core i7 Extreme	6/12	i7-3960X	
Core i7	6/12	i7-3930K	
Core i7	4/8	i7-3820	
Core i7	4/8	i7-27##K, i7-26##K, i7-26##, i7-26##S	
Core i5	4	i5-25##K, i5-25##, i5-25##S, i5-25##T, i5-24##P, i5-24##, i5-24##S, i5-23##P, i5-23##	Low-end[1]
Core i5	2/4	i5-23##T	Low-end[1][2]
Core i3	2/4	i3-21##T, i3-21##C, i3-21##	Low-end[1][2]

Pentium	2	G8##, G8##T, G6##, G6##T	Low-end[1][2]
Celeron	2	G5##, G5##T	Low-end[1][2]
Celeron	1/2	G47#, G46#	Low-end[1][2]
Celeron	1	G44#	Low-end[1][2]

Server

Family	Cores	Models	Known issues
Xeon E5	8/16	E5-4650, E5-4650L, E5-4640, E5-4620	Multi-CPU[3]; NUMA[4]
Xeon E5	8/16	E5-2687W, E5-2690, E5-2680, E5-2689, E5-2670, E5-2665, E5-2660, E5-2658, E5-2650, E5-2650L, E5-2648L	Multi-CPU[3]; NUMA[4]
Xeon E5	8/16	E5-2470, E5-2450, E5-2450L, E5-2448L, E5-2449L	Multi-CPU[3]; NUMA[4]
Xeon E5	6/12	E5-4610, E5-4607	Multi-CPU[3]; NUMA[4]
Xeon E5	6/12	E5-2667, E5-2640, E5-2630, E5-2620, E5-2630L, E5-2628L	Multi-CPU[3]; NUMA[4]
Xeon E5	6/12	E5-2440, E5-2430, E5-2420, E5-2430L, E5-2428L	Multi-CPU[3]; NUMA[4]
Xeon E5	6/12	E5-1660, E5-1650	
Xeon E5	6/12	E5-1356	
Xeon E5	6	E5-4617	Multi-CPU[3]; NUMA[4]
Xeon E5	4/8	E5-4603	Multi-CPU[3]; NUMA[4]
Xeon E5	4/8	E5-2643, E5-2618L	Multi-CPU[3]; NUMA[4]
Xeon E5	4/8	E5-2418L	Multi-CPU[3]; NUMA[4]
Xeon E5	4/8	E5-1620	

Xeon E5	4/8	E5-1410	
Xeon E3	4/8	E3-1290	
Xeon E5	4	E5-2609, E5-2603	Multi-CPU[3]; NUMA[4]
Xeon E5	4	E5-2407, E5-2403	Multi-CPU[3]; NUMA[4]
Xeon E5	4	E5-1607, E5-1603	
Xeon E5	2/4	E5-2637	Low-end[2]; Multi-CPU[3]; NUMA[4]
Xeon E3	4/8	E3-1280, E3-1275, E3-1270, E3-1260L, E3-1245, E3-1240, E3-1235, E3-1230	
Xeon E3	4	E3-1225, E3-1220	
Xeon E3	4/8	E3-1125C, E3-1105C	
Xeon E3	2/4	E3-1220L	
Pentium	2/4	350	Low-end[1][2]
Pentium	2	1407, 1405, 1403	Low-end[1][2]

Mobile

Family	Cores	Models	Known issues
Core i7 Extreme	4/8	i7-29##XM	
Core i7	4/8	i7-28##QM, i7-27##QM, i7-27##QE, i7-26##QM, i7-26##QM	
Core i7	2/4	i7-26##M, i7-26##LE, i7-26##UE	Low-end[2]
Core i5	2/4	i5-25##M, i5-24##M, i5-25##E	Low-end[1][2]
Core i3	2/4	i3-23##M, i3-23##E, i3-23##UE	Low-end[1][2]
Pentium	2/4	B9##C	Low-end[1][2]

Pentium	2	9##, B9##	Low-end[1][2]
Celeron	2	B8##, B8##E, 88#, 87#, 86#, 85#, 84#, 84#E	Low-end[1][2]
Celeron	1/2	80#, 7##C	Low-end[1][2]
Celeron	1	82#E, 7##, B7##, 80#UE	Low-end[1][2]

[1]**. The system is below the recommended hardware requirements**
This system is below the recommended hardware requirements.
The recommended CPU families are Intel Core i7, Intel Core i9, Intel Xeon.

[2]**. The system is below the recommended hardware requirements**
This system is below the recommended hardware requirements.
The recommended minimum amount of physical cores is 4.

[3]**. Multi-CPU usage**
Systems with multiple CPUs aren't supported by Reconstructor. In order to use the software with no performance penalty on such systems, the process affinity must be configured, so that it runs on all the cores of one single CPU. This will limit the amount of CPU cores and memory available to Reconstructor, but will allow multiple copies of the program to run on separate CPUs.

[4]**. NUMA configuration**
Systems that expose their CPU as multiple NUMA nodes aren't optimal for the heavy workloads Reconstructor is capable of. In order to run Reconstructor with no performance penalty caused by the memory partitioning, NUMA must be disabled on the computer's BIOS. Please contact your hardware manufacturer for details.

[5]**. Cluster-on-die configuration**
This family of CPUs offers the Cluster-on-Die functionality, that can split the CPU into multiple logical CPUs as clusters of cores, with separate RAM partitions. If this feature is enabled, the performance of applications requiring to share large memory allocation between multiple threads is negatively affected. In order to run Reconstructor without performance loss, it is recommended that the Cluster-on-Die functionality is disabled in the computer's BIOS. Please contact your hardware manufacturer for details.

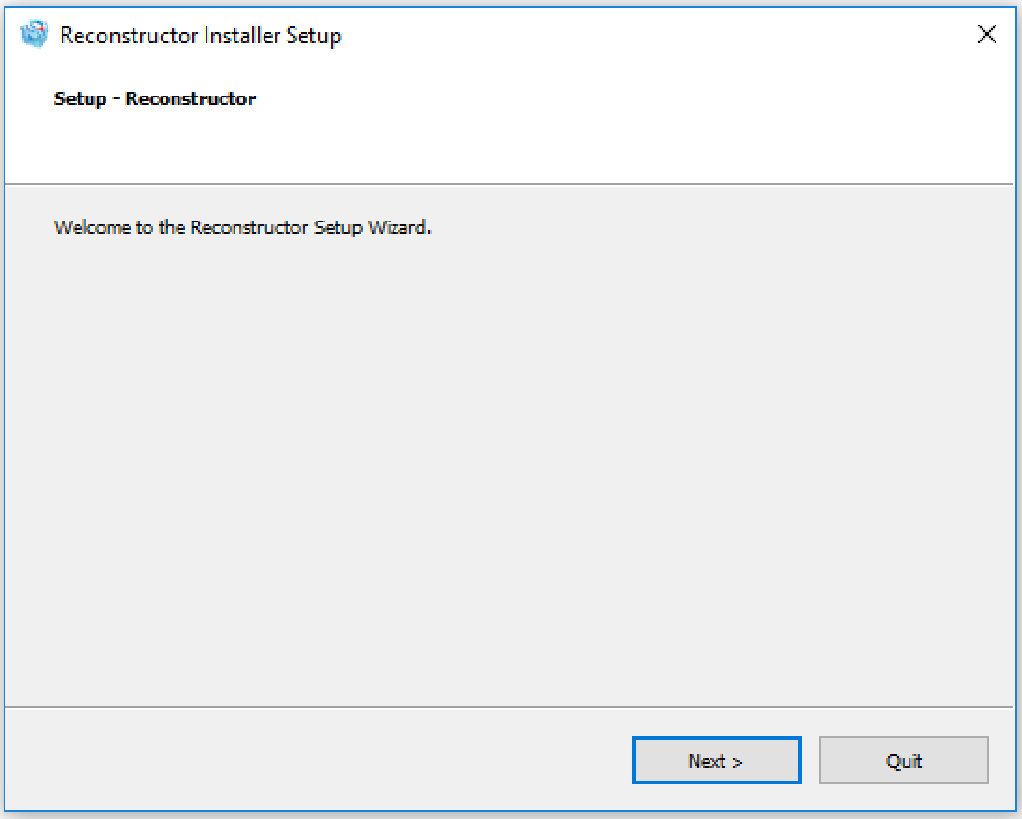
[6]**. Internal die topology**
This family of CPUs provides several cores in the same package, with multiple memory controllers shared by them. Access to memory across different CPU cores might suffer increased latency in some cases, and has not been benchmarked. On CPUs that support the Cluster-on-Die functionality, this can likely be mitigated by disabling such functionality, but further tests may be necessary.

Installation

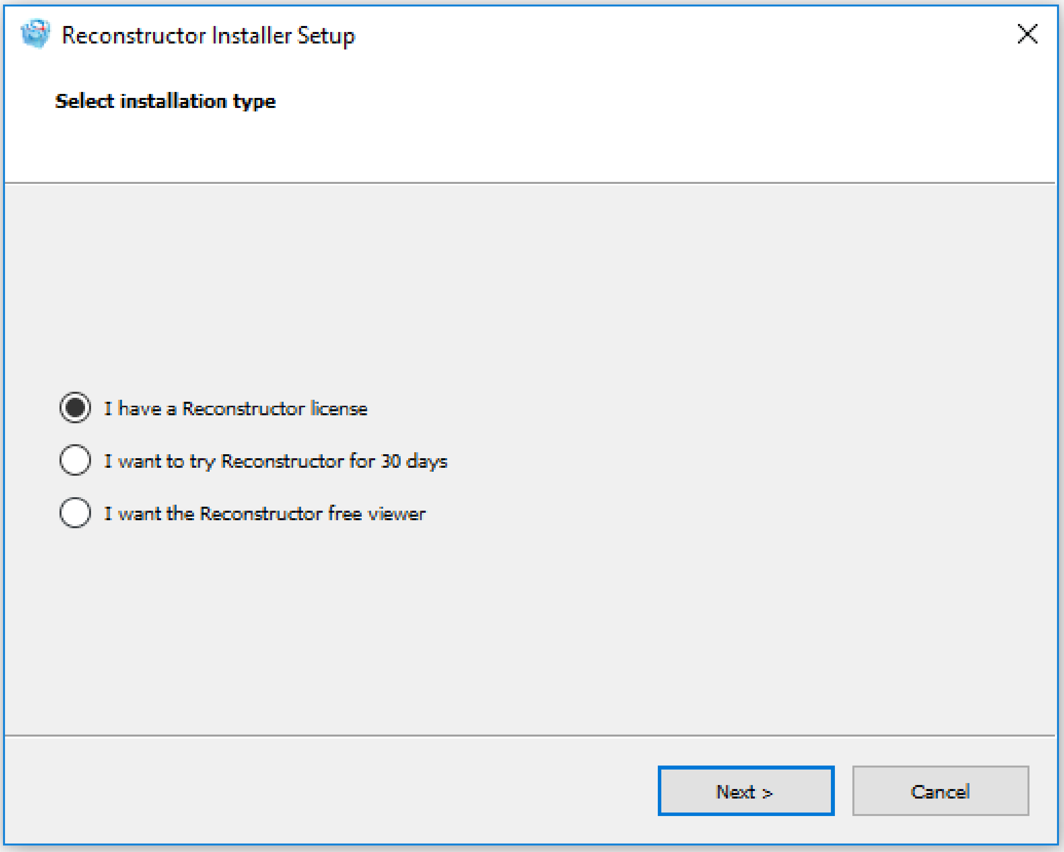
It's highly recommended both to deactivate all the antivirus and antispyware software and to keep closed any Reconstructor® application before starting the installation. Windows® default antivirus and firewall can remain active.

1. Run *Reconstructor_4.x.x.xxx_STONEX.exe*, press *YES* at the Windows alert message and follow the steps.

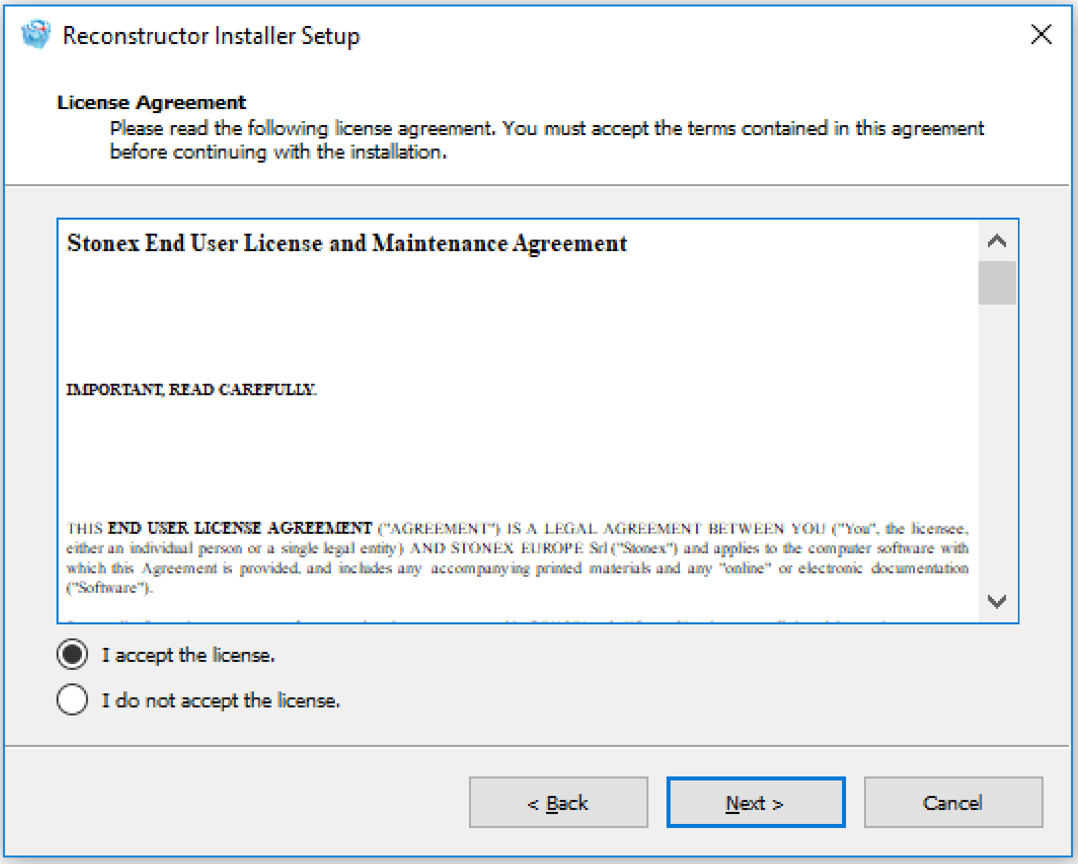
2. The Reconstructor Setup Wizard starts, press *Next* to continue...



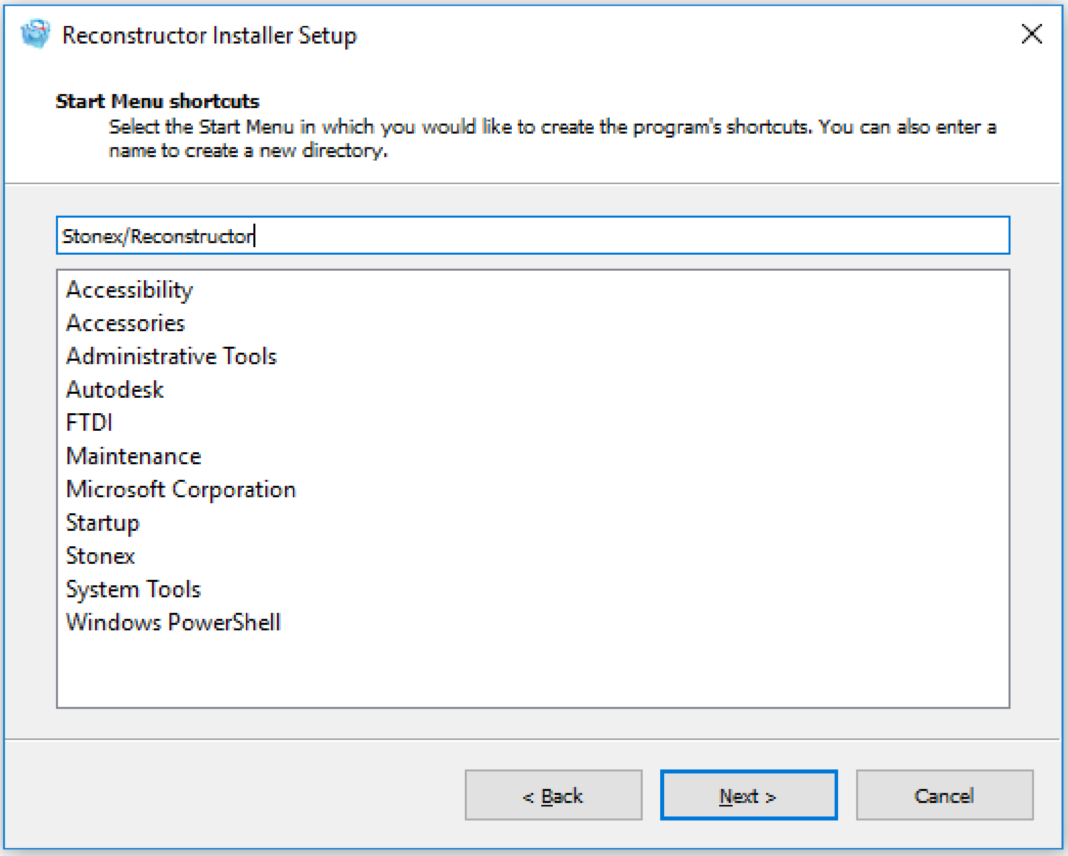
3. Choose the type of installation.



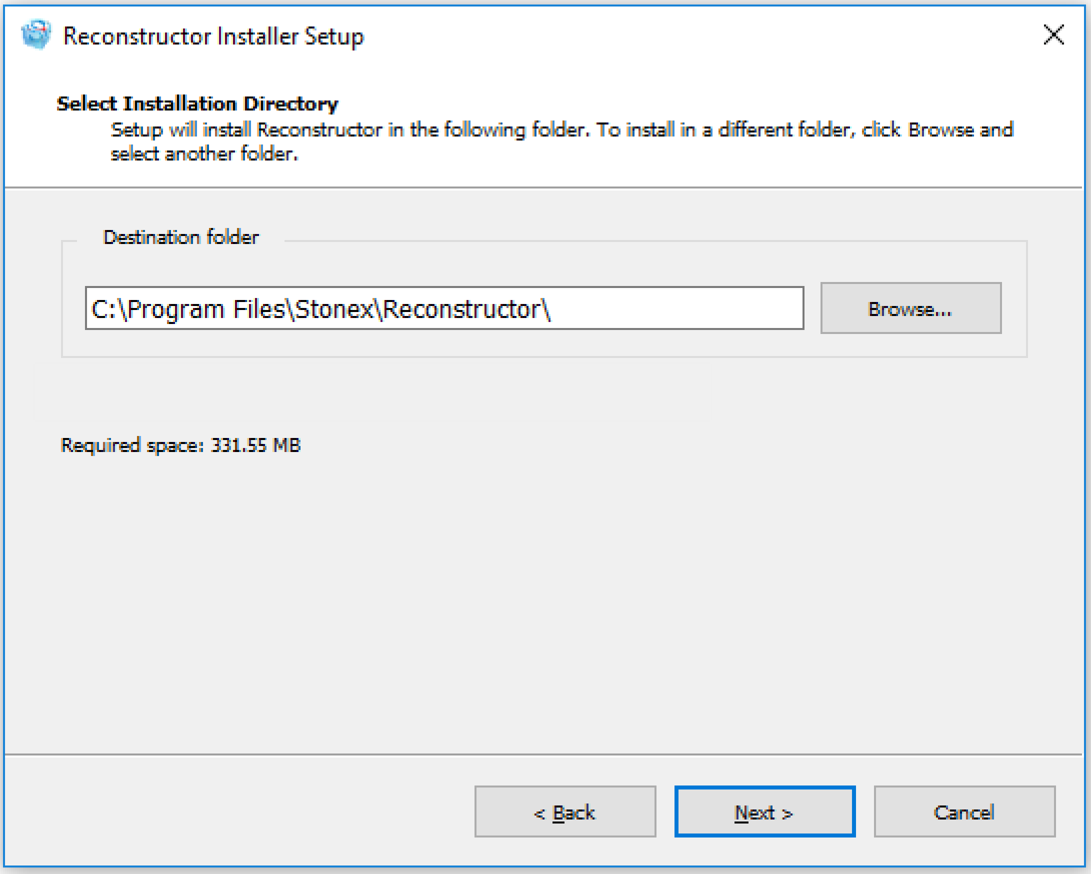
4. Accept the License Agreement terms and follow the steps.



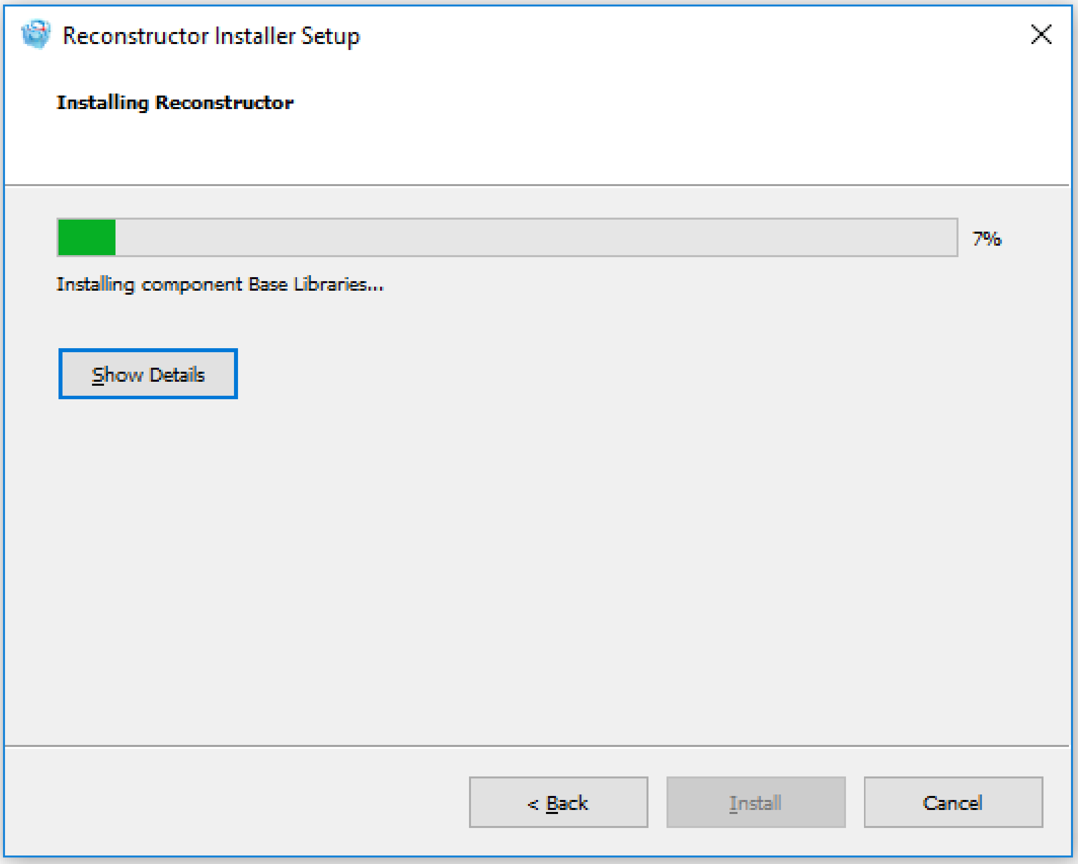
5. Choose Start Menu Folder shortcut and follow the steps.



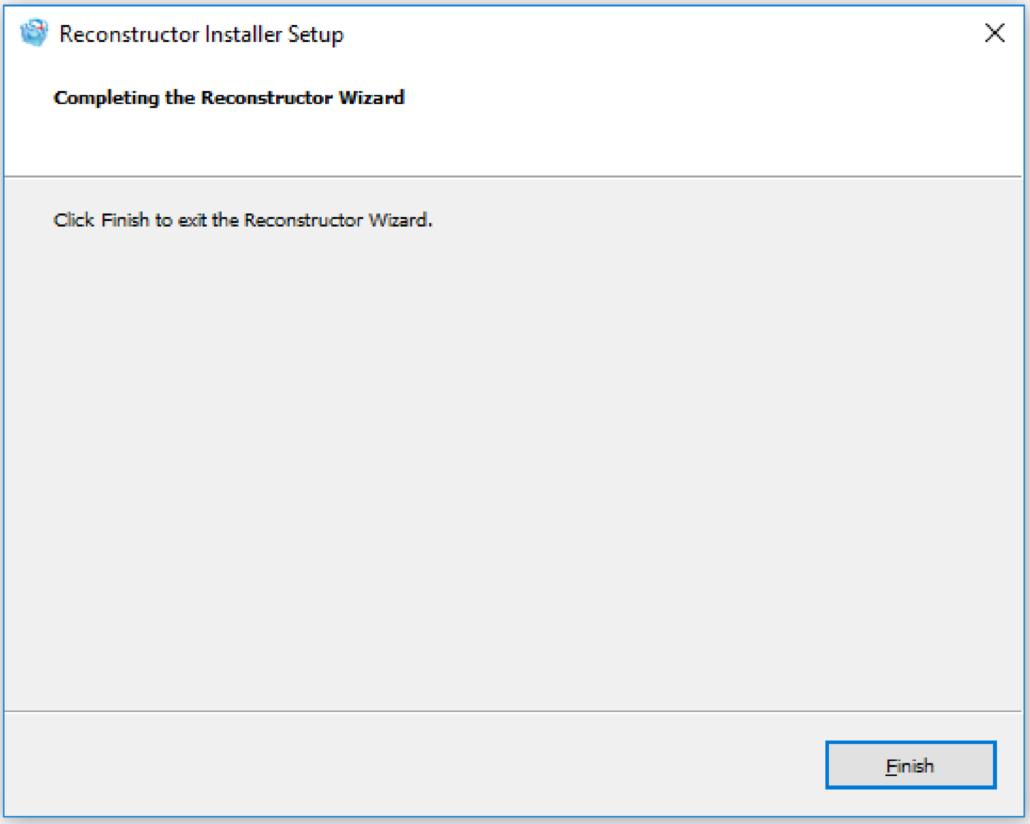
6. Choose Install Location (Default path is *C:\Program Files\Stonex\Reconstructor*).
   If a version of Reconstructor has already been installed, the installation procedure will uninstall the old one and then install the new one.
   Press *Next*...



7. Installation starting ...



8. At the end press *Next* and *Finish.*



9. This shortcut icon 

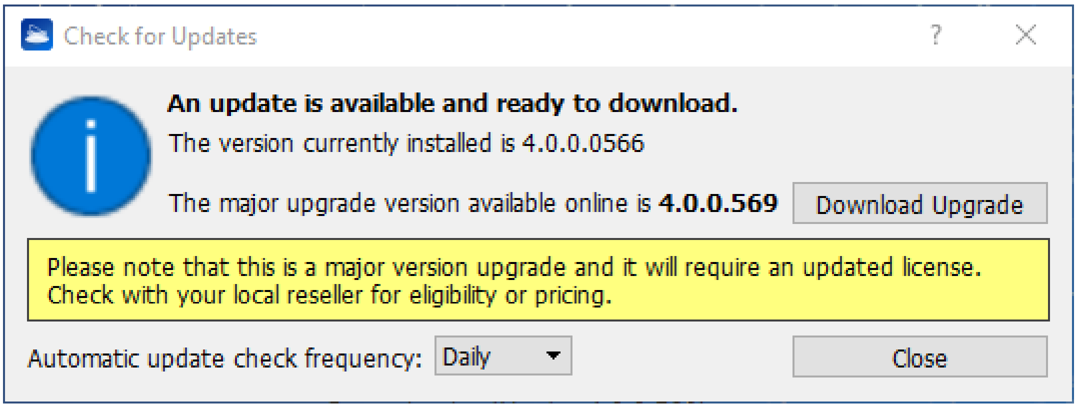
will appear on your desktop.

See also *Help* → *About Reconstructor®* to find all the information about the license, expiration date included.

Multi-licensing (USB server licenses)

To use the multi-seat licenses:
1. Install the same version of Reconstructor® on all computers to be used by selecting the type of installation "*I want to try Reconstructor for 30 days*". After installing the software reboot the PC before to continue.
The computers can be positioned in different locations, but they must be connected to the same network (same IP address and subnet mask).
2. Insert the USB key into one of the computers.
3. Once opened Reconstructor on a computer, the license will be deducted from the multi-seat license.

Reconstructor® automatically check new updates when available.


Check for Updates
An update is available and ready to download.
The version currently installed is 4.0.0.0566
The major upgrade version available online is 4.0.0.569
Download Upgrade
Please note that this is a major version upgrade and it will require an updated license.
Check with your local reseller for eligibility or pricing.
Automatic update check frequency:
Daily
Close

Licensing

Activation and updating software are performed in two different ways, depending on the kind of license.

USB (hardware) LICENSE
The Reconstructor® license is stored in a USB hardware key (dongle).
A USB license permits to move the dongle between several computers on which the software is installed.

COMPUTER BASED (digital) LICENSE
The Reconstructor® digital license key is closely linked to the computer used to generate the activation request file.

The first activation and the updating of a Reconstructor® license follow the same procedure:

Step 1 - Generating a **.c2v** file (to send to Stonex)

Step 2 - Applying a **.v2c** file (got from Stonex)

USB LICENSE

If you have a Reconstructor® USB dongle, how can you activate/update a license on it?

First of all, remember that each dongle is identified by its own ID, written on a label (see in the image GX0000 as example).
Please use this code when communicating with Stonex.

Step 1 - Request

1. Plug in the USB dongle
2. Open Reconstructor® and select *Help → License Manager* in order to create a .c2v request file.
3. Click on *Generate license activation request (C2V)*.
4. Choose the type of license key: *→ USB license (1 found)*
   The USB dongle is found, checked, and used to generate the request file .c2v.
5. Type the desired file name (for example *YYMMDD_DongleID_CompanyName*) and *Save* the .c2v file.
6. Send the .c2v file opening a ticket on MyStonex, or asking to your local reseller to do it for you.

Generating the activation file is possible from any PC.

Step 2 - Activation

After receiving the .C2V file, Stonex will send an activation .V2C file in reply.

Please, follow these instructions to apply it.

1. Plug in the USB dongle
2. Open Reconstructor® and select *Help → License Manager*.
3. Click on *Apply license activation file (V2C)*.
4. Select the .v2c file and *Open* it.
5. A successful message should appear at the end of the license activation process.
6. Reboot Reconstructor®.

Multi-licensing (only for USB licenses)

To activate a Multi- seats license on a USB server dongle remember to install Reconstructor® on all computers to be used by selecting the type of installation "*I want to try Reconstructor for 30 days*".
After installing the software reboot the PC before to continue.
See more details at Installation.

COMPUTER BASED LICENSE

If you want to activate/update a Reconstructor® license on your PC, how can you do?

Step 1 - Request

1. Install Reconstructor® on your PC, open it, and select *Help → License Manager* in order to create a .c2v request file.
2. Click on *Generate license activation request (C2V)*.
3. Choose the type of license key: *→ New computer based license/Computer based license (1 or more found)* [1]
4. Type the desired file name (for example *YYMMDD_CompanyName*) and *Save* the .c2v file.
5. Send the .c2v file opening a ticket on MyStonex, or asking to your local reseller to do it for you.

[1] *New computer based license* appears when no digital licenses were previously activated on the current computer and you need to request a digital key license for the first time

Computer based license (1 found) appears when a digital license was previously activated on the current computer
Computer based license (2 or more found) appears when more than one digital license was previously activated on the current computer. In this case select the first SL (software license) of the list to select the most recent

The digital license key is dependent on the computer used to generate the request activation file.

Step 2 - Activation

After receiving the .C2V file, Stonex will send an activation .V2C file in reply.

Please, follow these instructions to apply it.

1. Open Reconstructor® and select *Help → License Manager*.
2. Click on *Apply license activation file (V2C)*.
3. Select the .v2c file and *Open* it.
4. A successful message should appear at the end of the license activation process.
5. Reboot Reconstructor®.

⚠ For license transfer from one computer to another, please contact directly Stonex.

Disclaimer

GEXCEL rights reserved

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.
While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

All information and content (text, graphics and images) reported no sources are, to the best of our knowledge, in the public domain; if, unintentionally, copyrighted or infringing material has been published to the law please inform us and we will immediately remove it.

See also End User License Agreement.

Other right reserved

All trademarks shown belong to their legitimate owners; third party brands, product names, trade names, names corporations and companies (ex. Autodesk® , AutoCAD® , Revit® , ReCap® , Orbit® , Orbit GS® ...) mentioned may be trademarks of their respective owners or registered trademarks of other companies and they have been used for the sole purpose of explanation and for the benefit of the owner, without any purpose of violating the rights of Copyright in force.

Stonex End User License and Maintenance Agreement

IMPORTANT, READ CAREFULLY.

THIS END USER LICENSE AGREEMENT ("AGREEMENT") IS A LEGAL AGREEMENT BETWEEN YOU ("You", the licensee, either an individual person or a single legal entity) AND STONEX EUROPE Srl ("Stonex") and applies to the computer software with which this Agreement is provided, and includes any accompanying printed materials and any "online" or electronic documentation ("Software").
Stonex distribute the computer software under the agreement with GEXCEL srl ("Gexcel") who own all the right on the computer software.
All the conditions in agreement are dependent from agreement between Gexcel and Stonex.

This Agreement will also apply to any Software error corrections, updates and upgrades subsequently furnished by Stonex, unless such are accompanied by different license terms and conditions which will govern their use.

BY CLICKING THE "I ACCEPT" BUTTON, OR BY INSTALLING, COPYING OR OTHERWISE USING THE SOFTWARE, YOU AGREE TO BE BOUND BY THE TERMS OF THIS AGREEMENT. IF YOU DO NOT AGREE TO THE TERMS OF THIS AGREEMENT, YOU MAY NOT USE THIS SOFTWARE.

Stonex, previous Gexcel agreement may change the terms of this Agreement from time to time (such updated terms shall be uploaded onto Gexcel website at www.gexcel.it). Your continued access to and use of the Software will be deemed acceptance of the updated or amended terms. If you do not agree to the changes, you should cease using the Software.

This Software is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The Software is licensed, not sold.

1. SOFTWARE LICENSE.

1.1. License Grant. Subject to Your compliance with the terms of this Agreement, and any applicable restrictions set forth on the Purchase Order, Stonex grants You a non-exclusive, non-transferable license, without right of sublicense, to install, use and display the Software in a machine-readable form solely for Your internal purposes, and solely in connection with Your use of the Software. The Software installation is provided on a digital media (CD, DVD) or as a downloadable file (ISO image or executable file) from the Stonex online repository, whose link is communicated by e-mail. You may reasonably copy the Software to the extent necessary to enable Your permitted internal use of the Software, but You may not copy or distribute the Software to any third parties.

1.2. Other Rights and Limitations. (1) The Software contains valuable trade secrets proprietary to Gexcel and its suppliers or licensors. To the extent permitted by relevant law, You shall not, nor allow any third party to modify, adapt, translate, prepare derivative works from, copy, decompile, disassemble or otherwise reverse engineer the Software or any internal data files generated by the Software, or attempt to do so, provided, however, that to the extent any applicable mandatory laws give You the right to perform any of the aforementioned activities without Stonex's consent, You hereby agree that, before exercising any such rights, You shall first request such information from Stonex in writing detailing the purpose for which You need the information. Only if and after Stonex, under Gexcel agreement, at Gexcel's sole discretion, partly or completely denies Your request, may You exercise such statutory rights. (2) This Software is licensed as a single product. You may not separate its component parts for use on more than one computer or device. (3) You may not rent, lease, lend, or otherwise transfer rights to the Software or the use thereof to any third party. (4) Concurrent use of the Software is not permitted if not included in the license type.

1.3. Ownership Notices. (1) All right, title and interest (including but not limited to copyright, patent, trade secret and all other intellectual property and proprietary rights worldwide) in and to the Software (including but not limited to any images, photographs, animations, video, and text incorporated into the Software), the accompanying printed materials, and any copies of the Software are owned by Stonex and its suppliers or licensors. You shall not remove, cover or alter any of Gexcel's (or its designated suppliers' and licensors') copyright, trademark or other proprietary notices placed upon, embedded in or displayed by the Software or on its packaging and related materials. (2) You acknowledge that the Software is designed to work with, and contains, certain third party software, including without limitation, certain software licensed by its owners under open source software license (collectively, the "Third Party Software"). Any Third Party Software, whether distributed with the Software or otherwise obtained by You, is subject to the terms of the third party software license accompanying or otherwise applicable to that Third Party Software ("Third Party License"). The terms of the applicable Third Party License will apply to the Third Party Software independent of the terms of this Agreement. Nothing in this Agreement limits Your rights under, or grants rights to You that supersede the terms of any applicable Third Party License.

1.4. U.S. Government Restricted Rights. The software and documentation are "commercial items" as that term is defined at 48 C.F.R.2.101, consisting of "commercial computer software" and "commercial computer software documentation" as such terms are used in 48 C.F.R. 12.212. Consistent with 48 C.F.R. 12.212 and 48 C.F.R. 227.7202-1 through 227.7202-4, all U.S. Government end users acquire the software and documentation with only those restricted rights as set forth herein.

2. TERM.

2.1. Effectiveness. "Effective Date" means the date upon which You purchase a license Key from Stonex, or issues to Stonex a valid purchase order for a license Key. This Agreement is effective upon the Effective Date or, in any event, upon Your installation of the Software and continues unless terminated by Section 8 below.

2.2. Software Activation. The Software activation is provided by software-based protection keys ("Software Key") or hardware-based protection keys ("USB Dongle Key"). Following the receipt by Stonex of the applicable Fees for the relevant license, Stonex will issue to You a key to activate the Software for the license (a "Key"). You will be responsible for installing the Key to activate the Software. The Key shall be considered Confidential Information of Stonex, as that term is defined herein. Stonex shall provide temporal activation depending to purchase and payment conditions.

2.3. Evaluation Term. You are granted a limited right to use the Software, in compliance with this Agreement, free of charge for a period of evaluation commencing upon the date on which You install the Software. For details about the evaluation time please contact Laser.Team@stonex.it. Within the Evaluation Period, the Software runs as fully functional Trial except for the ability in exporting items.

3. FEES AND PAYMENT.

3.1. Fees. In consideration for the rights granted hereunder, You shall pay all the license Fees to Stonex in the amounts set forth on the Purchase Order on or before the Effective Date specified on the Purchase Order.

3.2. Terms of Payment. All payments are non-refundable You shall be responsible for all sales taxes, use taxes, withholding taxes, value added taxes and any other similar taxes imposed by any federal, state, provincial or local governmental entity on the transactions contemplated by this Agreement, excluding taxes based upon Stonex's net income. When Stonex has the legal obligation to pay or collect such taxes, the appropriate amount shall be invoiced to and paid by You unless You provide Stonex with a valid tax exemption certificate authorized by the appropriate taxing authority.

4. SOFTWARE MAINTENANCE.

4.1. Software Maintenance Services. During the maintenance term and subject to the terms and conditions hereafter specified, Stonex shall provide Software Maintenance Services, including both technical support services and software releases. (1) Technical Support Service is provided through Customer Support (2) As part of its maintenance services, Stonex may provide periodic Software

Releases, including Updates (generally available releases of Software that provide Bug Fixes, Error Corrections and Enhancements designated minor by Gexcel) and Upgrades (generally available releases of Software that provide enhancements designated major by Gexcel as well as minor new Enhancements and Error Corrections). Available Software releases, periodically issued by Gexcel, may either be downloaded from Stonex's web site or delivered on digital support upon written request to Stonex.

4.2. Software Maintenance Limitations. (1) Stonex shall have no obligation to provide Releases or Support for the Software except as set forth hereafter. Stonex shall not have any responsibility to develop subsequent components for the Software or additional processes for You, except as explicitly set forth herein. (2) Stonex shall have no obligation to fix errors in the Software within a specific time duration. Stonex will use commercially reasonable best efforts to fix bugs and other Software defects (but not including viruses or other defects not inherent in the Software when delivered By Stonex) in a timely manner during its normal business hours and upon reasonable notice, other than any bugs or defects caused by unauthorized use or modification of the Software, accident, abuse, misuse or misapplication of the Software or the magnetic or electronic media containing the Software. (3) You acknowledge that Stonex, after using all reasonable efforts, may not be able to resolve an Error and that such inability shall not be considered a breach or event of default under this Agreement, nor be the cause of any liability to Stonex. In the event that Stonex notifies to You its inability in fixing an Error in the Software, You may at your option and as your sole and exclusive remedy, terminate the Software Maintenance for the applicable Software. Upon such termination, neither party shall have any obligation or liability to the other in connection with such Software Maintenance. (4) Stonex shall have no obligation to fix Errors in any version of the Software other than the latest official release. (5) Your rights and obligations concerning the use of any Releases (Error Corrections, Enhancements, or any other programming provided by Stonex relating to the Software) shall be as provided under this End User License Agreement. (6) Maintenance Services do not include: (a) development of custom computer programs, (b) repairs or service relating to any Third Party Software or hardware, or (c) hardware modifications or changes to existing hardware configurations, outside the scope of warranty support. (7) Gexcel may, at its discretion, upon notice to You and with no additional charge, make modifications to the Software. Such modifications shall not jeopardize the basic functionality of the Software or invalidate this Agreement.

4.3. USB Dongle Key and Software Key Replacement. (1) For USB Dongle activation, within a period of ten (10) days from the date of acquisition, Stonex warrants that the USB Dongle Key will be free from defects in materials and faulty workmanship under normal use and service. If a defect in the USB Dongle Key should occur during such 10 days period, the USB Dongle Key may be returned to Stonex who will replace the USB Dongle Key without any charge for You, provided that the You have properly used the USB Dongle. After this period and if the failure of the USB Dongle Key has resulted from unauthorized use, accident, abuse, misuse or misapplication of the USB Dongle Key, You shall return toStonex the damaged dongle and pay to Stonex a Replacement Fee of 250,00 Euro (delivery charges excluded), as advised by Stonex. In case of USB Dongle Key loss, or stolen, You shall provide Stonex with a written and signed declaration and contact Stonex at Laser.Team@stonex.it. (2) The Software Key may be activated on a single computer system. In case of damages occurred to the installation computer system that do compromise the installed Software Key, You shall provide Stonex with a written and signed declaration and contact Stonex at Laser.Team@stonex.it.

4.4. Software Maintenance Term. The Software Maintenance Services provided under this Agreement shall run from the date of First License Activation for a time period of twelve (12) months or until the term in years for which the Maintenance Services have been purchased by You.

4.5. Software Maintenance Fee. The first Software Maintenance period of twelve (12) months after the First License Activation is available at a price included in the Software License Fee. The Renewal of Software Maintenance for the subsequent twelve (12) months may be available at a Renewal Fee of 15% of the full price of the purchased Software License, unless otherwise advised by Stonex.

5. NO WARRANTIES.

THE SOFTWARE IS BEING DELIVERED TO YOU "AS IS" AND STONEX MAKES NO WARRANTY AS TO ITS USE OR PERFORMANCE. STONEX AND ITS SUPPLIERS AND LICENSORS DO NOT WARRANT THE PERFORMANCE OR RESULTS YOU MAY OBTAIN BY USING THE SOFTWARE. YOU ASSUME THE ENTIRE RISK AS TO RESULTS AND PERFORMANCE OF THE SOFTWARE. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, STONEX AND ITS SUPPLIERS AND LICENSORS DISCLAIM ALL WARRANTIES, TERMS, AND CONDITIONS, EITHER EXPRESS OR IMPLIED, BY STATUTE, COMMON LAW OR OTHERWISE, INCLUDING BUT NOT LIMITED TO, IMPLIED WARRANTIES, TERMS, AND CONDITIONS OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, TITLE, AND NONINFRINGEMENT WITH REGARD TO THE SOFTWARE, ITS SATISFACTORY QUALITY, AND THE PROVISION OF OR FAILURE TO PROVIDE MAINTENANCE SERVICES.
STONEX IS NOT RESPONSIBLE FOR THE OPERATION OR FAILURE OF OPERATION OF THE LASER SCANNER OR ANY OTHER HARDWARE INCLUDED WITH OR RELATED TO THE INSTALLATION.

6. LIMITATION OF LIABILITY.

TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, IN NO EVENT SHALL STONEX OR ITS SUPPLIERS OR LICENSORS BE LIABLE FOR ANY LOST PROFITS OR COSTS OF PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES, OR FOR ANY SPECIAL, INCIDENTAL, INDIRECT OR CONSEQUENTIAL OR PUNITIVE DAMAGES, HOWEVER CAUSED AND REGARDLESS OF THE THEORY OF LIABILITY (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 OR INABILITY TO USE THE SOFTWARE, OR THE PROVISION OF OR FAILURE TO PROVIDE MAINTENANCE SERVICES, EVEN IF STONEX OR AN AUTHORISED STONEX'S REPRESENTATIVE HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES, AND NOTWITHSTANDING ANY FAILURE OF ESSENTIAL PURPOSE OF ANY EXCLUSIVE REMEDY PROVIDED IN THIS AGREEMENT.
IN NO EVENT SHALL STONEX'S TOTAL LIABILITY IN CONNECTION WITH THIS AGREEMENT AND THE SOFTWARE, WHETHER BASED ON CONTRACT, WARRANTY, TORT (INCLUDING NEGLIGENCE), STRICT LIABILITY OR OTHERWISE, EXCEED THE AMOUNT PAID TO STONEX FOR USE OF THE SOFTWARE GIVING RISE TO THE CLAIM, IF ANY. BECAUSE SOME STATES AND JURISDICTIONS DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL OR INCIDENTAL DAMAGES, THE ABOVE LIMITATION MAY NOT APPLY TO YOU.
STONEX SHALL HAVE NO LIABILITY TO YOU FOR ACTUAL DAMAGES FOR ANY CAUSE WHATSOEVER, REGARDLESS OF THE FORM OF THE ACTION.

7. CONFIDENTIALITY.
The term "Confidential Information" shall mean any information disclosed by either party (the "Disclosing Party") to the other party (the "Receiving Party") in connection with this EULA which is disclosed in writing, orally or by inspection and is identified as "Confidential" or "Proprietary." Notwithstanding the foregoing, the following is "Confidential Information" of Stonex: (a) any information, in whatever form, disclosed by Stonex that

relates to the Software and that is not publicly known; (b) any benchmark tests pertaining to the Software; and the Software license Key provided by Stonex to You hereunder. The Receiving Party shall treat as confidential all Confidential Information received from the Disclosing Party, shall not use such Confidential Information except as expressly permitted under this Agreement, and shall not disclose such Confidential Information to any third party without the Disclosing Party's prior written consent. The Receiving Party shall use the same degree of care to protect the Confidential Information of the Disclosing Party against unauthorized use or disclosure that it takes with its own confidential information of like nature, but in no event less than a reasonable degree of care. Notwithstanding the above, the restrictions of this Section shall not apply to information that: (a) was independently developed by the Receiving Party without any use of the Confidential Information of the Disclosing Party; (b) becomes known to the Receiving Party, without restriction, from a third party without breach of this Agreement and who had a right to disclose it; (c) was in the public domain at the time it was disclosed or becomes in the public domain through no act or omission of the Receiving Party; or (d) was rightfully known to the Receiving Party, without restriction, at the time of disclosure. Without terminating any restrictions or obligations of this Section, a Receiving Party is not prohibited hereunder from disclosing Confidential Information of the other party if such information is disclosed pursuant to the valid order or requirement of a court, administrative agency, or other governmental body of competent jurisdiction; provided, however, that the disclosure is strictly limited to the extent necessary to comply with such order and, provided further that, the Receiving Party shall provide prompt notice thereof to the Disclosing Party and shall use commercially reasonable efforts to obtain a protective order or otherwise prevent public disclosure of such information.

8. TERMINATION.

This Agreement is effective until terminated. Without prejudice as to any other of its rights, Stonex may terminate this Agreement immediately and without notice if You breach any provision of this Agreement or of any other agreement between You and Stonex. Upon termination or expiration of this Agreement: (a) any fees owed to Stonex under this Agreement before such termination or expiration will be immediately due and payable; (b) all licenses granted to You in this Agreement will immediately cease; and (c) You must promptly discontinue all use of the Software, uninstall and erase all copies of the Software from Your computers, and return or, at the option of Stonex, destroy all copies of the Software in Your possession or control. Sections 1.3, 1.4, 4, 5, 6, 7, 8 and 9 will survive termination or expiration of this Agreement for any reason.

9. EXPORT RESTRICTIONS.

You shall not export, directly or indirectly, the Software (or any component part or output thereof) to any country in violation of any law or regulation, including without limitation any law or regulation of the U.S. Government or any agency. You are solely responsible for complying with all applicable export and import regulations and obtaining all necessary export and import licenses or permits for the Software. You agree to defend, indemnify and hold Stonex harmless against any liability (including attorneys' fees) arising out of Your failure to comply with the terms of this paragraph.

10. USE OF CUSTOMER'S NAME.

You acknowledge that Stonex may use Your company name on Stonex's customer list provided to third parties.

11. GENERAL.

This Agreement shall be governed by the laws of Italy without regard to conflicts of law principles. The Courts of Italy will have exclusive jurisdiction over any disputes arising under this Agreement and the parties hereby irrevocably consent to the personal jurisdiction of such courts. In the event any proceeding or lawsuit is brought by the parties in connection with this Agreement, the prevailing party in such proceeding or lawsuit will be entitled to receive its costs and reasonable attorneys' fees. If any provision of this Agreement is held to be unenforceable, that provision will be modified so as to be enforceable, or if such modification is not possible, will be removed and the remaining provisions will remain in full force.
No failure or delay by either party in exercising any right, power, or remedy under this Agreement, except as specifically provided herein, shall operate as a waiver of any such right, power or remedy. Without limiting the foregoing, terms and conditions on any purchase orders or similar materials submitted by You to Stonex shall be of no force or effect. Neither this Agreement nor any rights or Your obligations under this Agreement may be assigned by You in whole or in part without the prior written consent of Stonex (except in connection with a merger or acquisition, or the sale of all or substantially all of such party's assets), and any attempted assignment or transfer by You in violation of the foregoing will be null and void. This Agreement, is the complete and exclusive statement of the agreement between the parties and supersedes any proposal or prior agreement, oral or written, and any other communications between the parties in relation to the subject matter of this Agreement. The parties agree that the United Nations Convention on Contracts for the International Sale of Goods is specifically excluded from application to this Agreement. This Agreement is in the English language only, which language will be controlling in all respects.

Getting Started

The aim of this section is to suggest a simply and fast workflow to work with Reconstructor®.

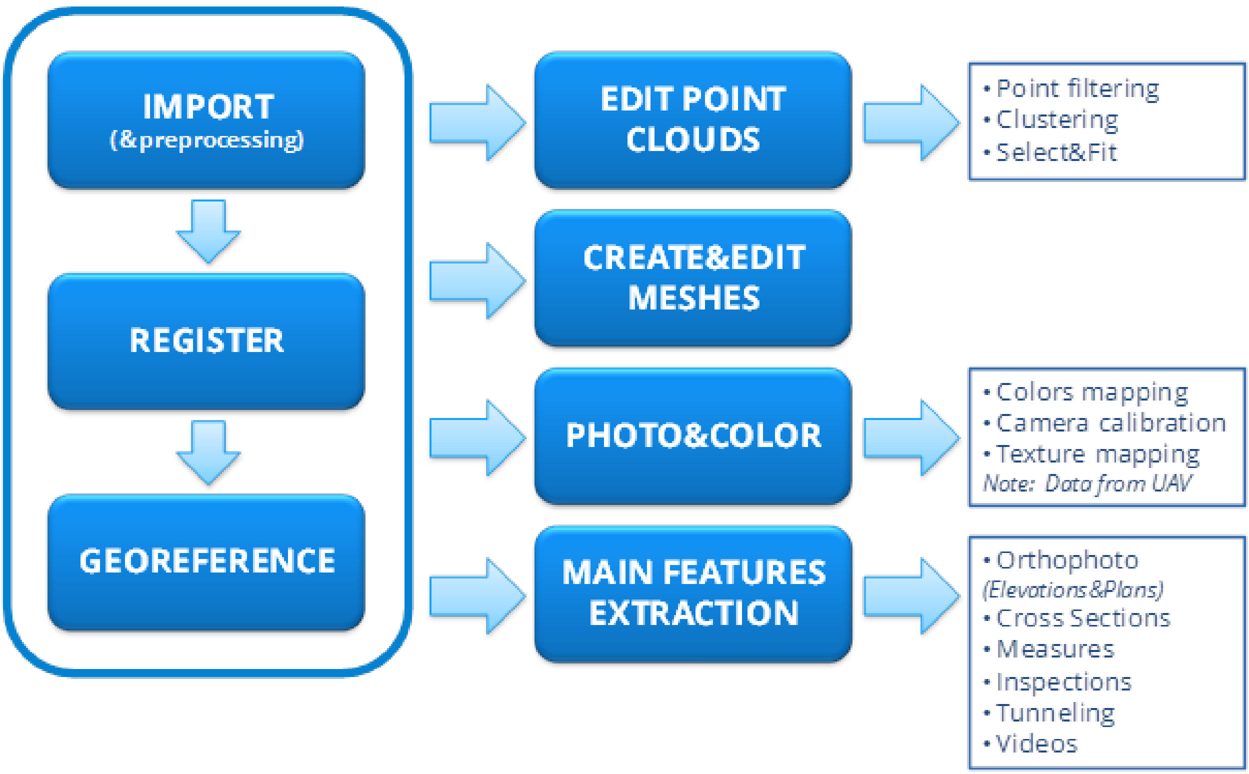
Before to use Reconstructor® you have to install the software.
After the installation the licensing permits must be verified and set.
See Licensing to learn how to request and apply your license.

At the opening of the software you need to Create a New project or Open an existing project (*).

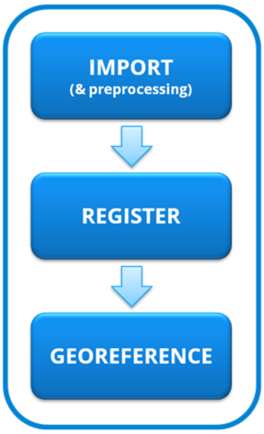
Each project has its own folder called with the same name. Reconstructor® projects contains at least some of the following subfolders: *Documents/ Results/ Imports/ Exports/ Unstructs/ Grids/ Meshes/ Polylines/ Trash/ Images/ Movies/*. These folders are automatically created when the project gets populated.

(*) Note: For major software releases, backward compatibility of Reconstructor projects is supported; forward compatibility is not supported.

Then, starting from importing, preprocessing, registering and georeferencing steps you can navigate through point clouds editing, meshing, coloring tools until the main features extraction.



The main Workflow



Most of the operations in the first part of the main workflow can be performed with LineUp® toolbar commands.

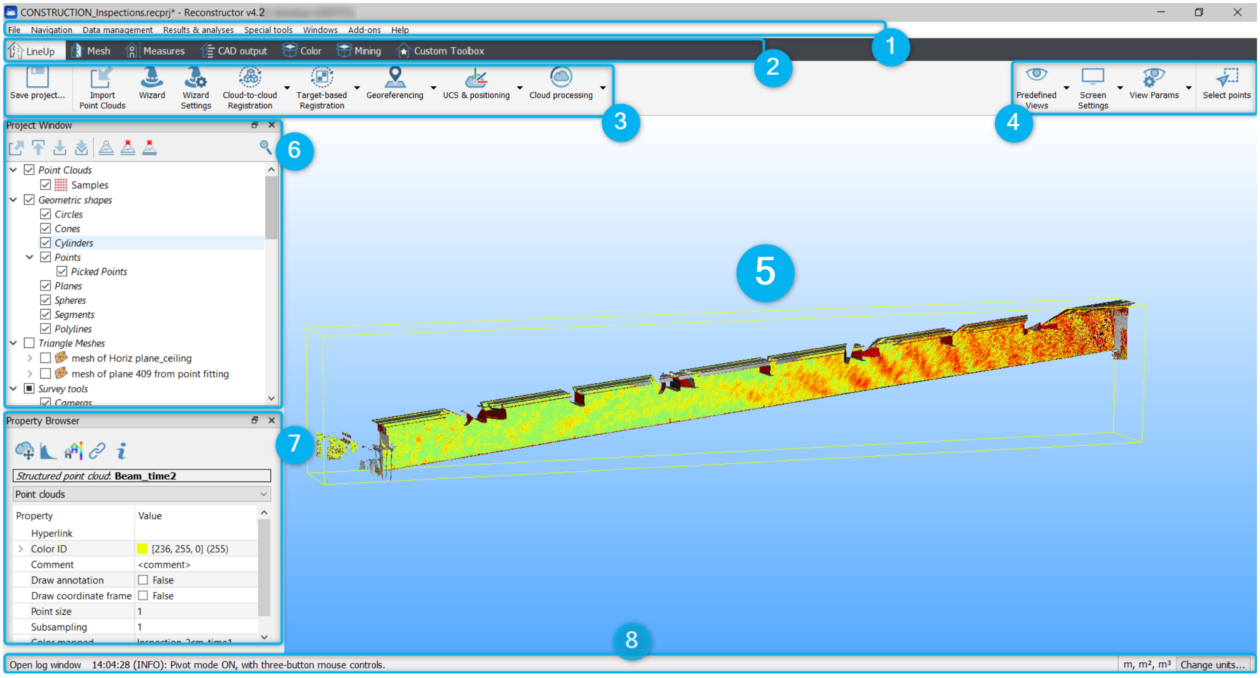
See at the below voices for all just mentioned processes:

1. Import
2. Register
3. Georeference
4. Edit Point Clouds
5. Create & Edit Meshes
6. Photo & Color
7. Results & Analyses

You can find all the Reconstructor commands in the *Command Dashboard* (Ctrl+d).

User Interface Layout

The Reconstructor® interface is simple and user-friendly, so most of the main commands are in the Top Toolbar.



The user interface is composed by:

1. The menu bar on the uppermost part of the window: contains all the command and software functions
2. The toolbox bar with the main topics
3. The top toolbar with shortcuts to the most useful and relevant commands for each of topic toolbox
4. The view and select tools with current view, rendering parameters and points select options
5. The main rendering window where you can view in 3D the project objects
6. The project window (dockable), which lists the objects that make the 3D scene (imported or created during the processing)
7. The property browser (dockable), which allows to view and edit the properties of the selected objects
8. The bottom toolbar composed by the log line, that shows what is happening (the button opens the log window), and the units of measure space (current shown, edit by button)

Furthermore, by right-clicking on an item in the project window, a context menu is displayed where only the commands that apply to the given item type are shown. Commands can therefore also be issued from that context menu: this is useful in particular to load/unload a model, to set an hyperlink to an item or to open the hyperlink, to move the item to another group in the project.

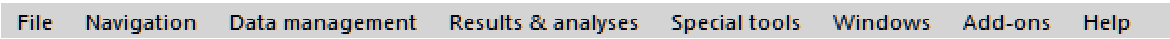
There are also short sentences (*tooltips*) that describe each project command; they can be shown simply by focusing on the chosen icon.

You can find all the Reconstructor commands in the *Command Dashboard* [Ctrl+D].

The Reconstructor® language can be managed from the GUI options tool.

Menu bar

The menu bar is located on top of Reconstructor®'s main window, and contains the commands and tools. Some of the commands are also accessible from the top toolbar and the contextual menu.



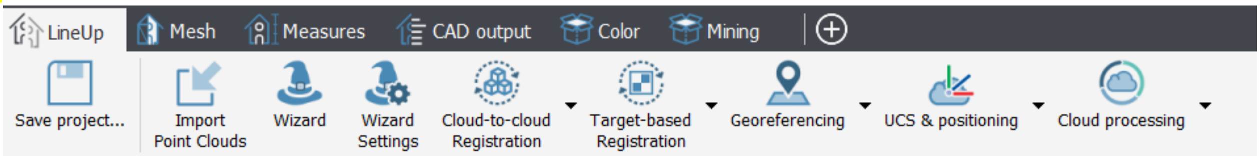
The menu bar features the following main voices:

- File: to import/export, load and save data and to manage the project
- Navigation: to interact with the 3D scene and manage rendering settings
- Data management: to register, process, edit your data
- Result & analyses: to extract results from the project data
- Special tools: auxiliary tools to process the data
- Windows: to manage the dockable windows
- Add-ons: to activate further specific functions
- Help: to access the online help, the license manager and to display information on the software version in use.

Please click on the blue words here above to study them in deep.

Toolbox Bar and Top Toolbar

Toolbox Bar and Top Toolbar allow fast access to frequently used commands from the File menu, Navigation menu and Outputs menu.
The top toolbar changes when the user changes the current Toolbox.



The tabs of the toolbox change according to the active Add-on:

- LineUp, Mesh, Measures and CAD output tabs are available in the Reconstructor basic version;
- Color and Mining tabs are available (and displayed) only if the related add-ons has been purchased.

The tools included in each tab are organized in order to group all the commands useful to process data in a particular field of application.
So don't be surprised if you find some commands present in more than one tab, because they are re-proposed to facilitate the workflow.

LineUp
To Import, Preprocess, Register and Georeference any set of points clouds (both unstructured and structured *-grid-* point clouds).

Mesh
To extract different kind of triangular mesh models and manage them. Also creation and management of geometric shapes are included.

Measures
To take measurements (distances, areas, volumes), annotations, to record videos and to run mesh-cloud comparison (inspection).

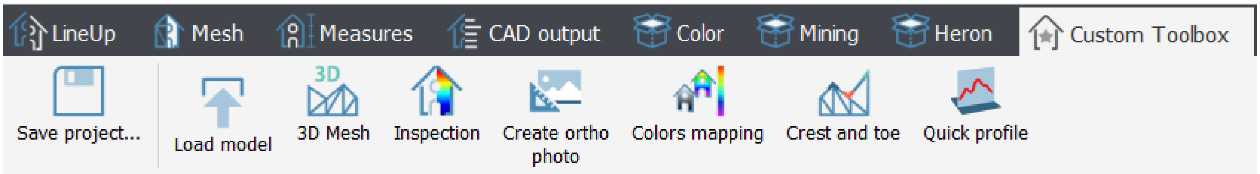
CAD output
To extract outputs that can be exported in CAD environment, as cross sections, quick profiles, orthophotos and X-ray orthophotos.

Color (only available with COLOR MAPPING add-on)
To manage all the levels of colorization processes, including colorization of point clouds with external high-resolution pictures, creation of textured meshes, management of projectors.

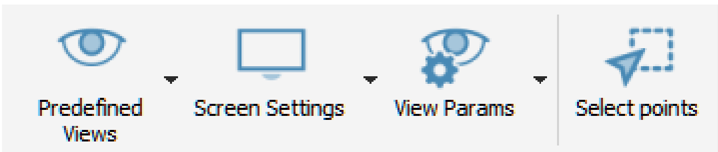
Mining (only available with MINING add-on)
To extract topographic meshes and DEM models, to compute volumes/cut&fill volumes and extract contours. Special tools for tunneling applications are included.

Custom Toolbox

It's also possible to activate the Custom Toolbox with custom tools chosen by the user.



View, Navigation and Select points tools



View tools

On the right of the Top Toolbar there are three commands containing the main functions related to the possible views, the screen settings and the parameters of the current view.

Predefined Views
Several 3D and 2D views are here available to visualize the models in the most functional way.

Screen Settings
To set different screen and display options.

View Params
To choose and manage Perspective or Orthographic views.

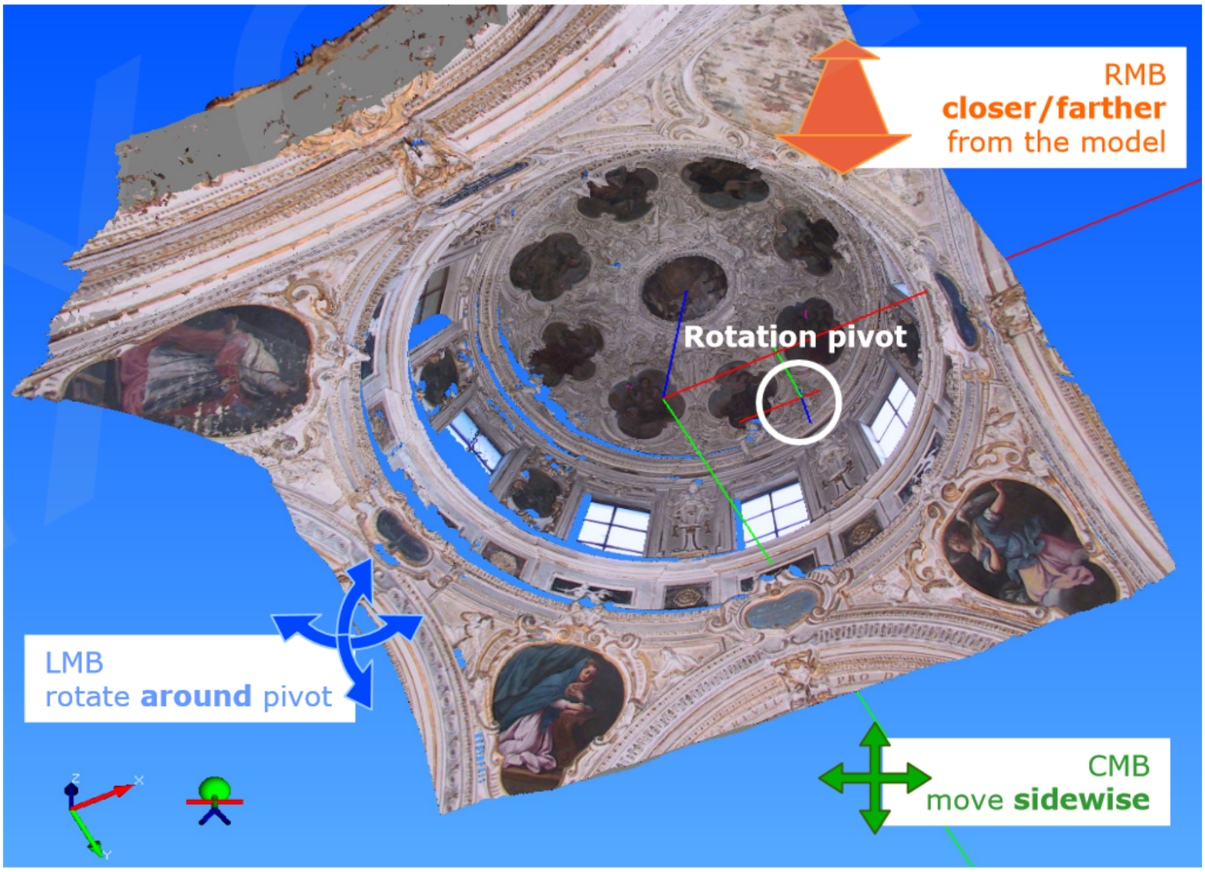
Navigation tools

Pivot Mode

Different Navigation modes are available; between them the *Pivot Mode* is the default and most used.

Pivot mode
Default navigation mode based on translating and rotating around a chosen point of the 3D scene

In this mode a point can be selected as a rotation pivot. The pivot is set by a left button click on a valid point of an object.



Keyboard/mouse movement	Effect
LMB + move mouse	Camera rotation around the pivot

CMB + move mouse	Camera sideways translation, pan
RMB + move mouse back/forth	Camera dollying, forward and backward along the current focal axis
Mouse wheel rotation	Translation towards point indicated by the mouse pointer
Shift + **LMB** + move mouse left/right	Camera rolling
LMB click on a model's point	Selection of rotation pivot
Shift + mouse wheel rotation	Zoom in and out (change of field of view)
Alt + **LMB** double click	Selection of points for point list window
Shift + **CMB** + mouse move	Draw rectangle to move into
LMB double click	Select the item

The *Bubble mode* may instead be useful to put yourself in the laser scan's point of view.

Bubble mode
Navigation mode in which only rotations around yourself are allowed

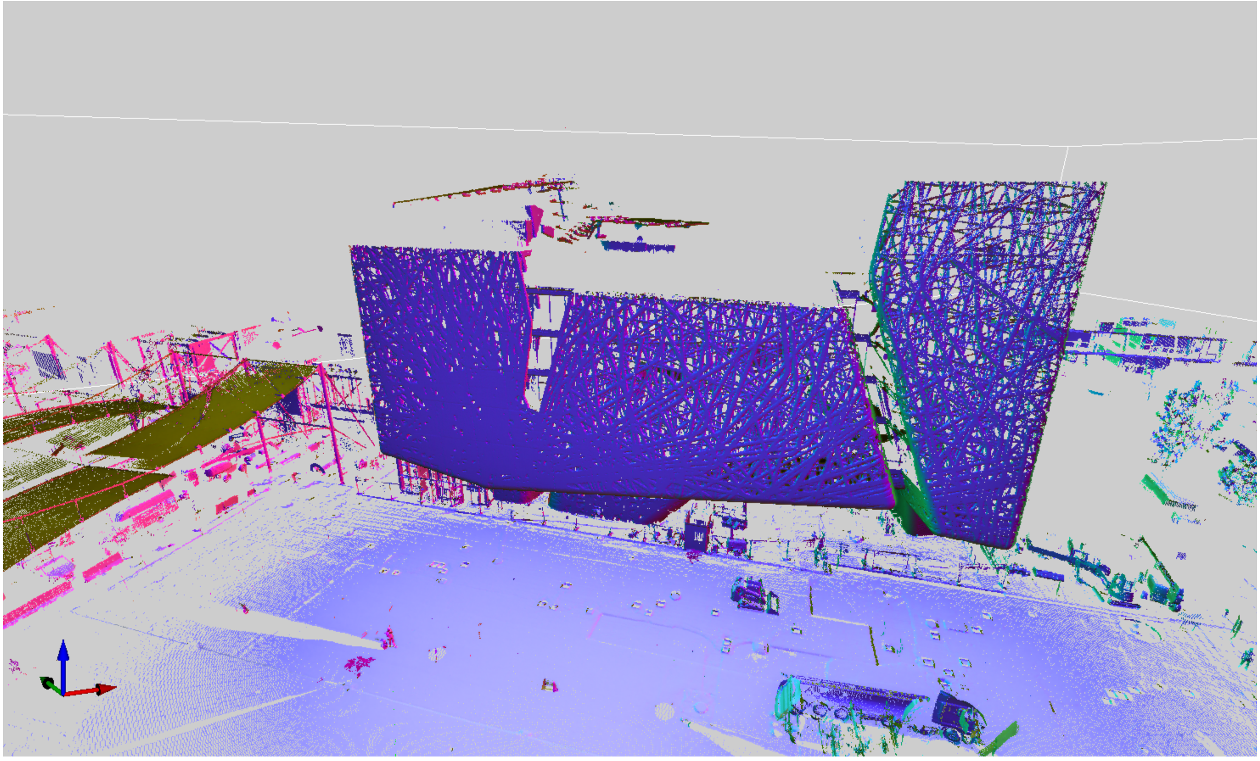
See the Navigation chapter for details and other navigation modes.

Select Points tools

The last command on the right of the Top Toolbar is dedicated to:

Select Points
This tool opens a recipe window to select, delete, hide and classify points.
A set of captured points can also be saved as a point cloud.

The 3D Viewport (or Main Rendering Window)



This window is located in the center of Reconstructor®'s GUI and it renders the 3D scene with all your project items.

In the screenshot above, you can see that the 3D window shows in the bottom left corner a triplet of coordinate axes. This triplet shows how the current UCS is oriented with respect to your current viewing direction.

On the right of the axes triplet, a small avatar is shown. If this avatar is present, it means that the system for human movements while rotating is on. This system is controlled through the Navigation options dialog, and it is explained also in that help page.

Tip: to easily add models to the scene, drag and drop your files onto the Reconstructor® window.

Navigation

Reconstructor® provides many interaction styles to navigate in the 3D scene.
See in Navigation menu to learn how to navigate in Reconstructor® environment.

Interaction with models

You can select models in this window by left-double-mouse-click. Use it with Ctrl key pressed to easily select more items!
The contextual menu is available by right-double-mouse-click on the item in the 3D viewport.
This is an alternative mode to select items in the Project Window.

Points Selection

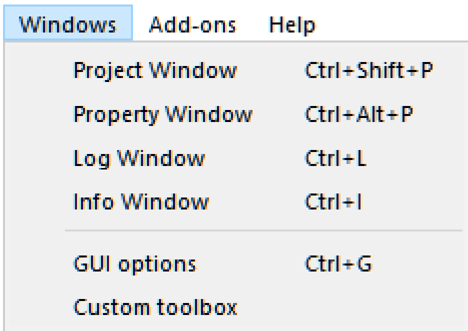
By pressing the space bar, the 3D window goes from *Navigation mode* to *Points Selection mode* and back. You can enter the selection mode by clicking *Tools->Selection&Fitting tools->Select* or by toggling the corresponding button on the top toolbar.
See Selection&Fitting tools for details.

Coordinate system

The renderer draws with a right-handed Cartesian coordinate system; see the XYZ badge in the bottom-left window corner, it represents the current UCS (User Coordinate System).

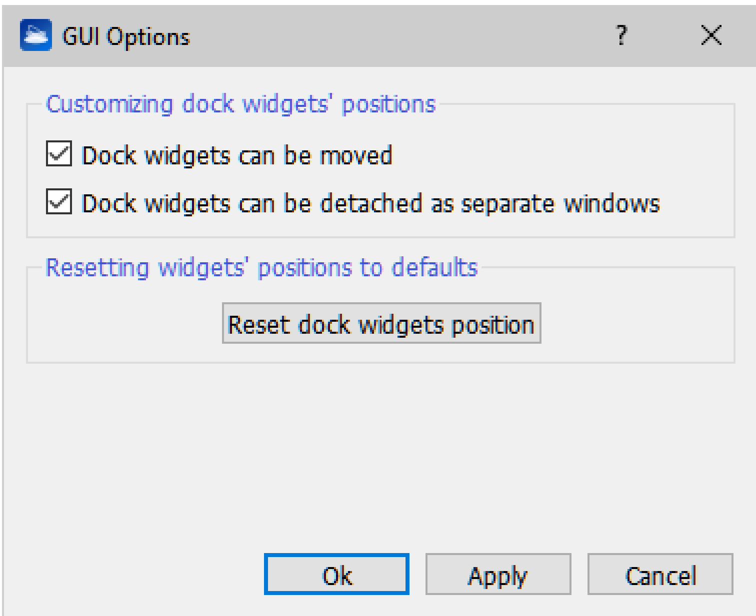
Dockable Windows

Reconstructor® GUI (Graphical User Interface) is composed also of many dockable windows. Each of these windows has a default docking place. However, the user can change it, move the windows and even leave them floating above the main GUI.



- *Project window*
- *Property window*
- *Log window*
- *Information window*
- *Recipe windows*

To reset the GUI to the default status open the *Windows* menu -> *GUI Options* and click on the *Reset dock widgets position* button.



The Recipe Window

This window is a particular dockable window, and its default docking area is on the right side. This window is automatically shown and hidden by Reconstructor® at need. It is shown for the time needed by a particular procedure to interact with the user.

Therefore, the recipe window doesn't have a fixed content, but hosts a particular dialog for the time that this is needed to the users. Here is a list of dialogs that are hosted in the recipe window:

- *Points selection with polyline*
- *Mesh from predefined view*
- *Volume calculation*
- *Area of cloud portion*
- *Area of mesh portion*
- *2D mesh selection*
- *3D mesh selection*
- *Cut and fill calculation*
- *Topographic meshing*
- *Plane creator and editor*
- *Create Orthophoto*
- *Create X-Ray Orthophoto*

Project Items

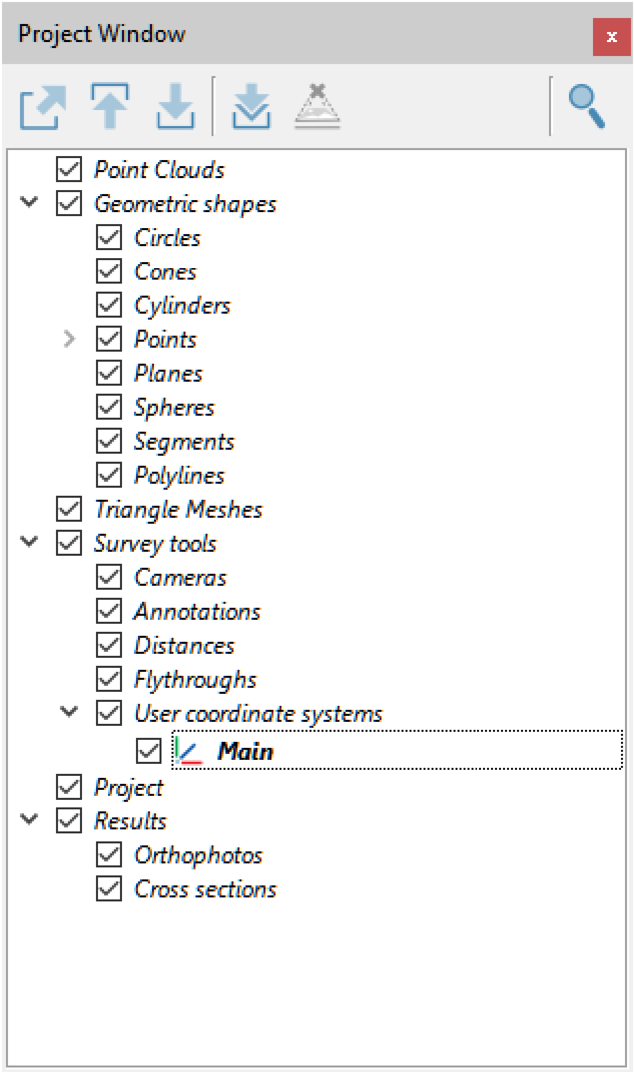
Reconstructor®'s 3D world can be populated with several kinds of items. Items are listed in a tree view in the project window.

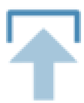
- **Point Clouds**: a a list of Point Clouds (both structured and unstructured)
- **Triangle Meshes**: a list of meshes, where each mesh is a collection of triangles with adjacency information
- **Geometric shapes**
  Primitives (planes, cones, cylinders, spheres, circles, segments) created by a robust fitting algorithms, polylines (created by contour algorithms or cross sections), picked on 3D scene or imported points
- **Survey tools**
  Cameras (preferred view points saved by the user with a definable projection among orthographic, perspective, cylindrical and spherical)
- **Annotations**: user comments attached to valid 3D points in the scene. Distance and angle quotations are considered as annotations
- **Distances**: obtained by distance measurements
- **Flythroughs**: spline trajectories defined by the user in order to create animations and movies
- **User Coordinate Systems**: the user can create different coordinate systems with special purposes
- **Project**: useful to organize and merge in a single location all the items of a sub-project
- **Results**: Orthophotos, Cross Sections , Quick profiles, Contours an Tunneling results are here listed.

  Section from plane, Contours, Quick profile and Tunnel cross sections tools give as output a *group item composed by one orthophoto, one polyline and one slice of point cloud for each extracted section. See more details in* *Cross sections structure overview*.

The Project Window

All the item groups are yet listed in this *Project Window* of a new clean project:





Load model:
select a model (cloud of points or mesh) from the project window tree and press this button, the model will be loaded in PC RAM and if it is also checked ☑, it will appear in the 3D viewport.

Unload model:
this button unloads from the RAM an from the main window the selected item(s).

Export model as:
depending on the type of object selected, an export window will appear; there are many formats available, see here.

Search project item:
searches through the items of the project.

Contextual menu functions

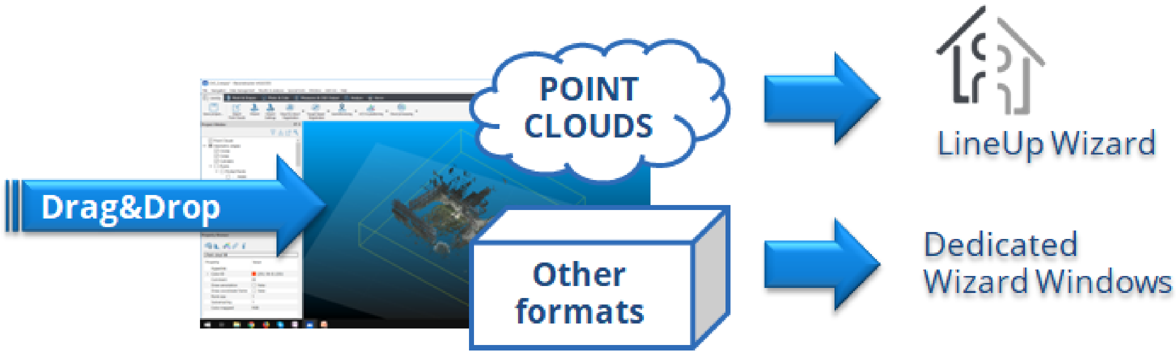
The contextual menu contains, among other functions:

- **Load/Unload model**: each model can be loaded or unloaded to free main memory
- **Save**: this command is available only if the data or properties has been modified
- **Save copy as**: save a copy of the model or export to another file format. The following dialog allows to browse for the file location and format (Files of type combo box)
- **Go to**: center the 3D window to the bounding box of the model. The view point is computed so that to contain the whole bounding box in the viewport. Warning: this could place sometimes the model far away, with the effect that it seems not visible. Therefore increase the max depth in the Options.
- **Center to local origin**: the view point is aligned to local coordinate frame of the model, with X rightwards, Y upwards and Z going out of the screen.
- **Align to bounding box**: the view point is aligned to the desired face of the bounding box of the model
- **Pose**: open the pose dialog to view and modify the transformation matrix and geo-referentiation of the model
- **{Model specific operations}**: list of specific processing functions.

See also project window dedicated chapter.

Import

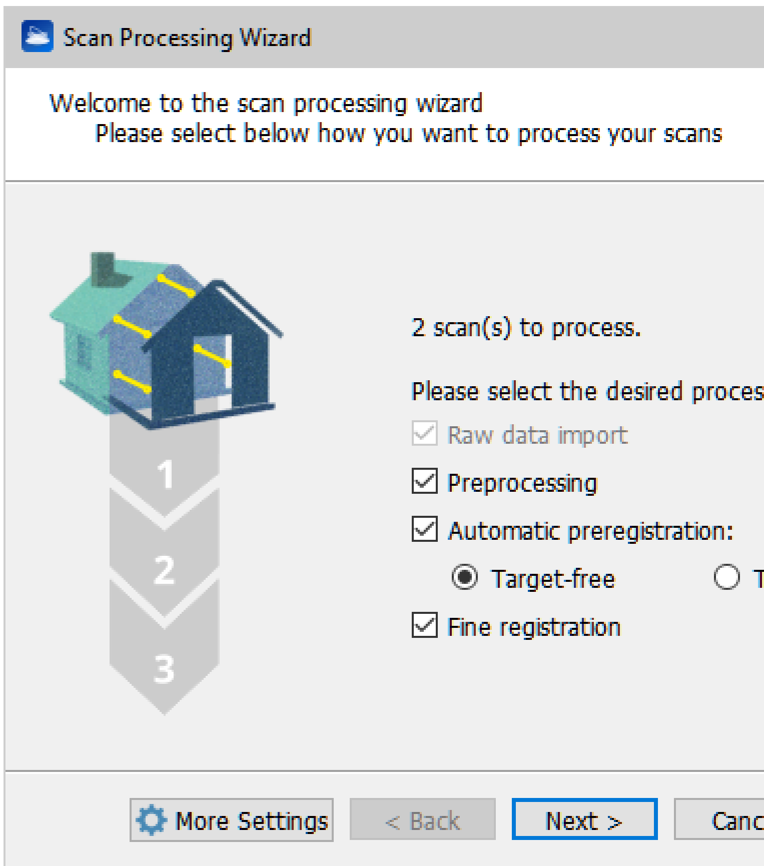
To quickly import data, just drag and drop the files you want to import anywhere in the 3D scene. A wizard will appear to ask you for import options, depending on kind of data.



a. If you're importing a point cloud the LineUp® Scan Processing Wizard will be opened.
b. If you're importing meshes and CAD models a specific window will appear to guide you through importing steps.
b. If you're importing a different file format (as polylines, scan projects, etc...) a specific window will appear to guide you through importing steps.

a. Importing Point Clouds... & Preprocessing

If you're opening one or more point clouds by drag&drop (or by *Import Point Clouds* command) a LineUp® Scan Processing Wizard will appear.



By default you can follow wizard instructions step by step, in order to import, preprocess and register the scans, but if you only want import raw data you have to uncheck *Preprocessing*, *Automatic preregistration* and *Fine registration*.

More information about point clouds import formats here.
More information about LineUp® Scan Processing Wizard here.

b. Importing Meshes and CAD Models

- If you're importing one or more of these objects:
    - Triangle meshes
    - CAD Models

a specific window will guide you in the importing step (advanced settings are used for IFC files).

c. Importing "Other Formats"

If you're importing one or more of these objects:

- Polylines
- Trajectories
- Specific images

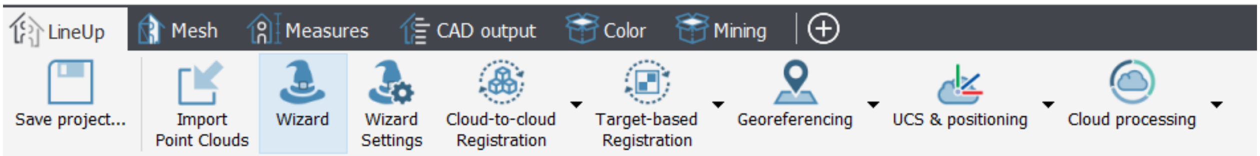
a specific window will guide you in the importing step (advanced settings are used for IFC files).

At the end of both of these import phases, your models will be loaded in your project and saved in Reconstructor® internal format.

See details in File > Import section.

See also the *Registration* step to know how align scans and meshes.

LineUp®



LineUp® toolbox easily allows you to:

- Import
- Preprocess
- Register
- Georeference

any set of points clouds (both unstructured and structured *-grid-* point clouds).

Import Point Clouds
To import any point cloud format.

Wizard
Import, preprocess, preregister (with or without targets) and fine register single or multiple scans together.

Wizard Settings
Define advanced settings for the scans processing workflow.

Cloud-to-cloud Registration
Manual and Automatic preregistration and registration commands

Target-based Registration
Manage a register using targets

Georeferencing

UCS & positioning

Cloud processing

See complete guide here: LineUp.

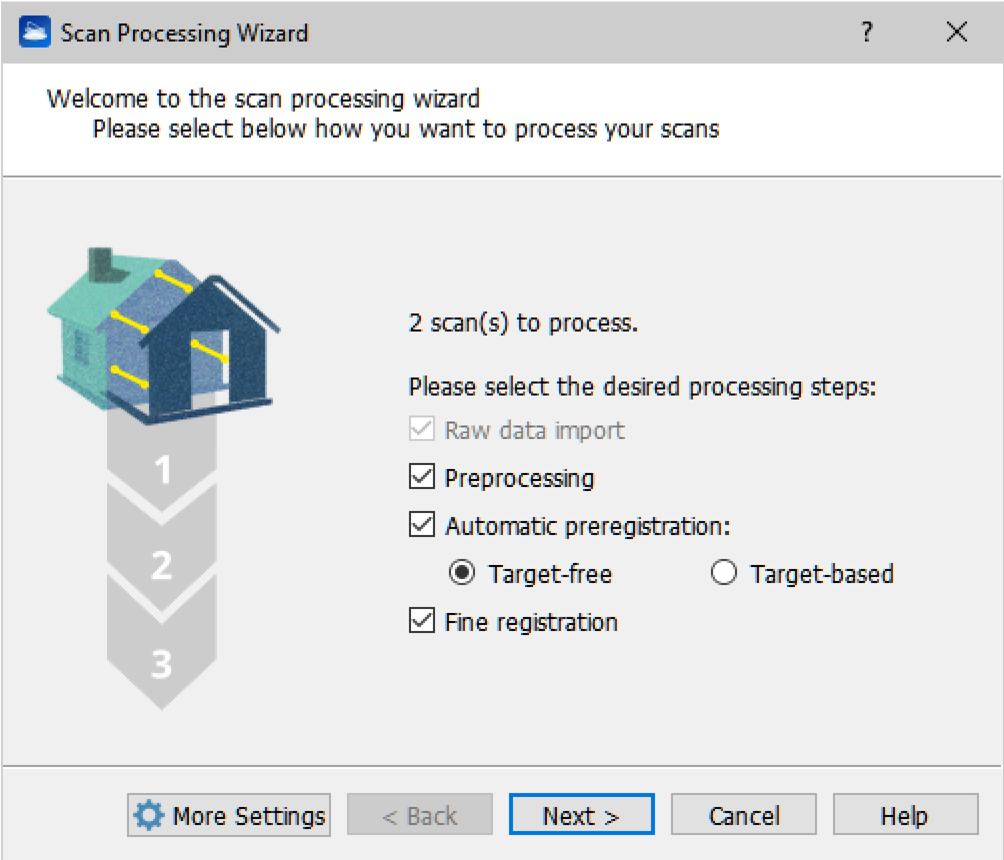
See *Register* step to know how align the scans.

Import Point Clouds

To quickly import data, just drag and drop the files you want to import anywhere in the 3D scene. A wizard will appear to ask you for import options.

Otherwise you can use the *Import Point Clouds* button in the LineUp top toolbar.

If you're opening one or more point clouds by drag&drop (or by *Import Point Clouds* command) a LineUp® Scan Processing Wizard will appear.

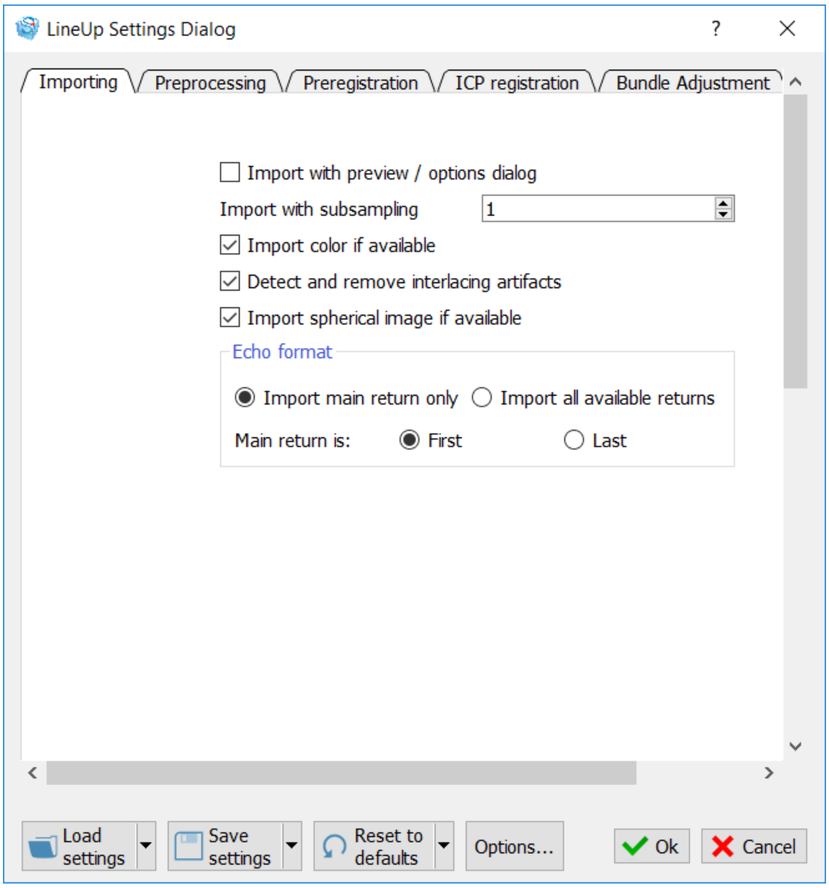


By default you can follow wizard instructions step by step, in order to import, preprocess and register the scans, but if you only want import raw data you have to uncheck *Preprocessing*, *Automatic preregistration* and *Fine registration*.

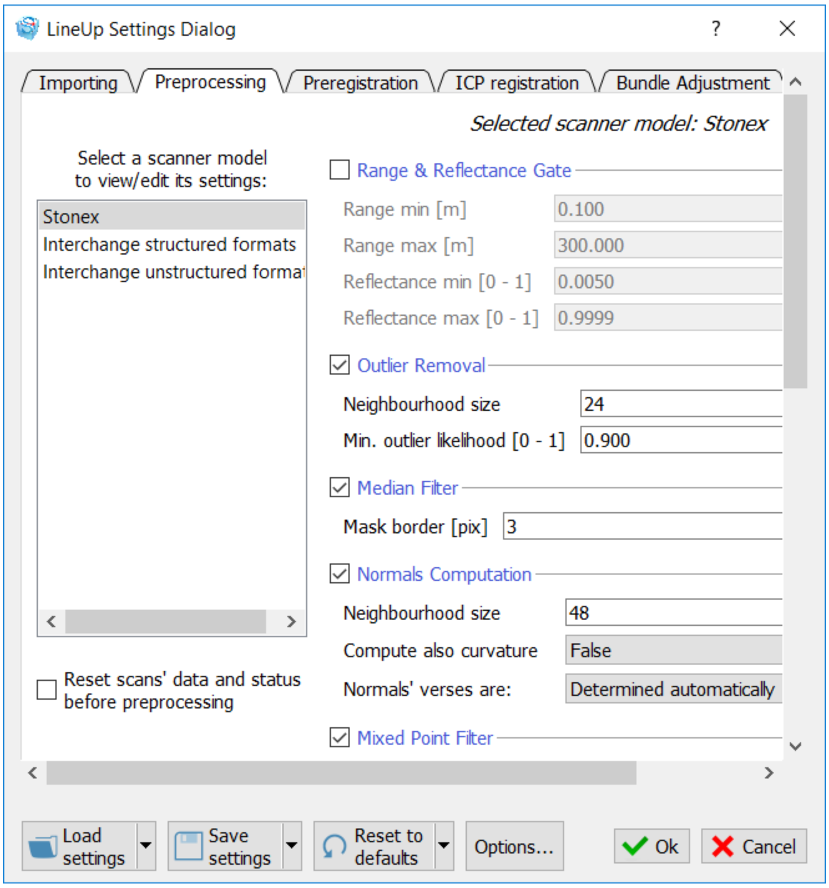
More information about point clouds import formats here.
More information about LineUp® Scan Processing Wizard here.

Clicking on *More Settings* button a *LineUp Settings Dialog* will appear.

In this phase you have to pay attention to *Importing* and *Preprocessing* folders:



You have simply to choose the subsampling rate. Depending on your data format you can evaluate several import options, as color or multi echoes.



Start LEVEL

You have only to identify your data import format (look at your sensor) and use the default parameters to preprocess (clean and filter) the scans.

Advanced LEVEL

Depending on your data format and on your survey you can evaluate several options.

After closing this setting dialog click on *Next* and then on *Process* button.

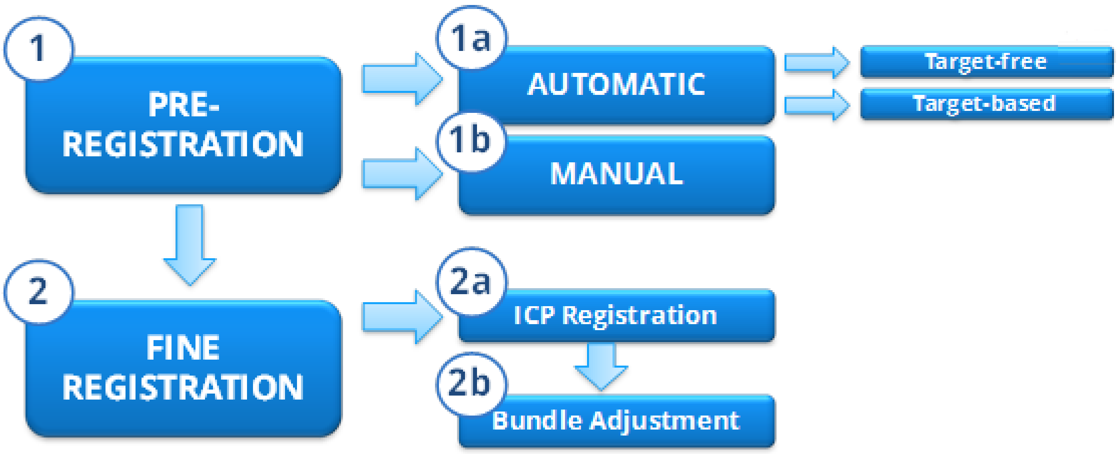
Import formats
Several formats for Unstructured and gridded point clouds can be imported:

Comma Separated Values (*.csv)
DEM ASCII header+matrix (*.asc)
E57 grid point cloud (*.e57)
Generic binary traster (*.*)
Grid Point Cloud Text File (*.txt)
PTX (*.ptx)
Stonex multiple scans (*.x3m)
Stonex point cloud (*.x3s)
E57 unstructured point cloud (*.e57)
LAS format (*.las)
LAZ format (*.laz)
PLY point cloud (*.ply)
Unstructured Point Cloud Text File (*.txt)
PTS (*.pts)

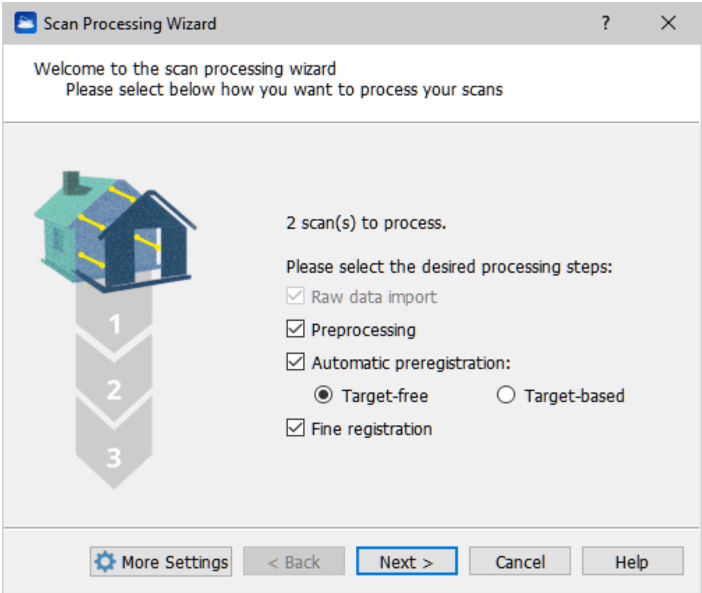
Registration

The LineUp® toolbox includes a comprehensive suite of tools for coarse and fine registration of any amount of scans (and not only). The registration process is composed by several steps, depending on adopted survey techniques.

The general flow, here illustrated



starts from a preregistration step that allows you to compute a rough alignment between the imported models. The alignment can be later refined using ICP registration and Bundle Adjustment to choose and refine the good ICPs and discard the wrong ones, in order to reduce the global registration error.

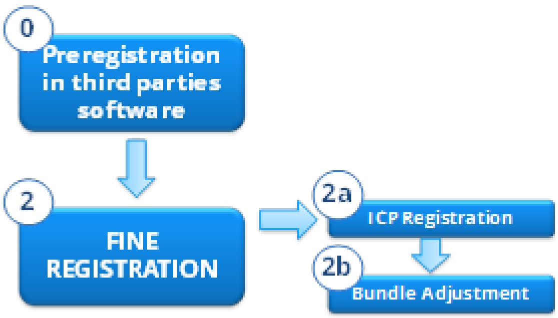


Start LEVEL You can simply use the *Scan Processing Wizard* (also simultaneously with the import phase) and check all the steps, using default parameters.

Advanced LEVEL You can manage the registration parameters for your purpose in the *LineUp Setting Dialog* or you can separately carry out the processes (with *Register* commands in the LineUp® main window).

In the following paragraph several customized workflows are illustrated. The user can choose the way, depending both on survey techniques adopted and on desired output accuracy. Then the main processes (algorithms) are briefly illustrated.

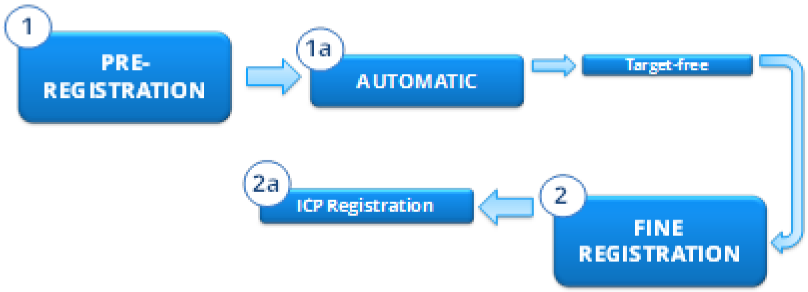
WF1 After importation of a pre-registered project



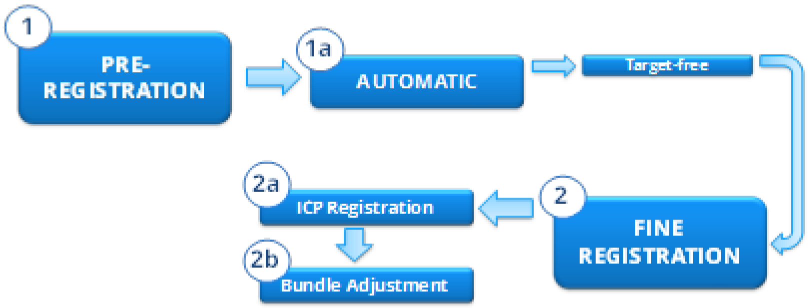
If you pre-registered your scans with a third part software (as the owner laser scan) you can import the just aligned scans and simply make a fine registration, in order to further reduce the registration error.

WF2 Automatic registration without targets

If you made a survey not using targets or markers, with a good overlapping (the minimum overlap of two scans is 20-30% of their surface), the automatic preregistration algorithm permits you to register all the point clouds you want.
Then you can decide if you want only to make a cloud to cloud registration

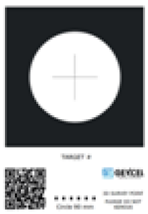


or also a bundle adjustment

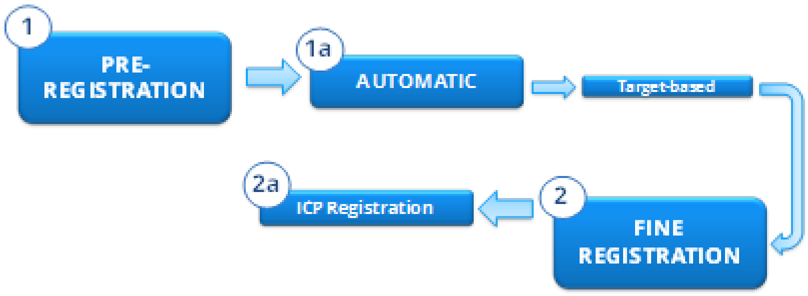


WF3 Automatic registration using targets

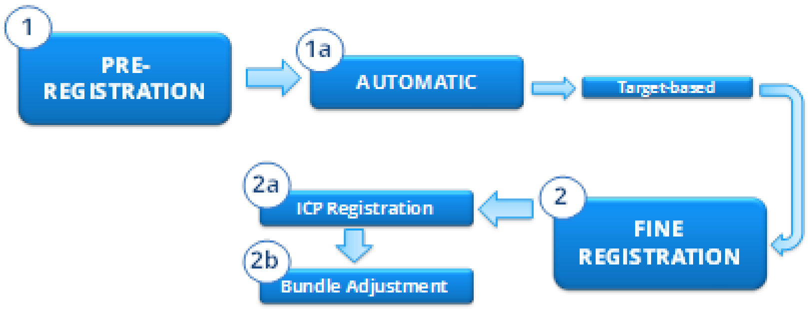
If you made a survey using Stonex's targets, a special function automatically register a set of scans by detecting and matching targets.



Then you can decide if you want only to make a cloud to cloud registration



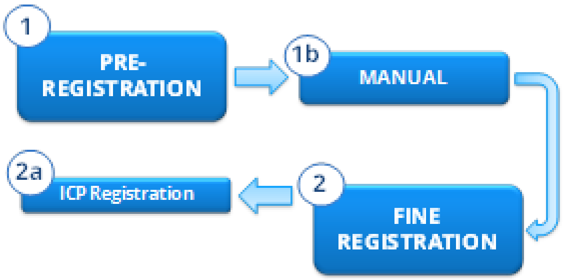
or also a bundle adjustment



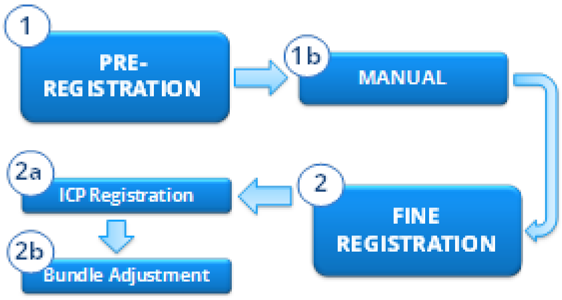
WF4 Automatic preregistration failed? Make a manual preregistration

If you don't achieve a good error of alignment using automatic techniques, the manual registration helps you to align two scans (or two generic models) by finding 3 couples of corresponding points among the models.

Then you can decide if you want only to make a cloud to cloud registration



or also a bundle adjustment



You can also choose to follow an alignment per groups procedure.

See in the following links how to implement:

1. Preregistration
2. Fine registration

See Georeference step to know how georeference the registered models.

1. Preregistration

1a. Automatic Preregistration

Target-free

This function is a very effective and fast procedure that preregisters a set of scans automatically, without using targets or markers.

Target-based

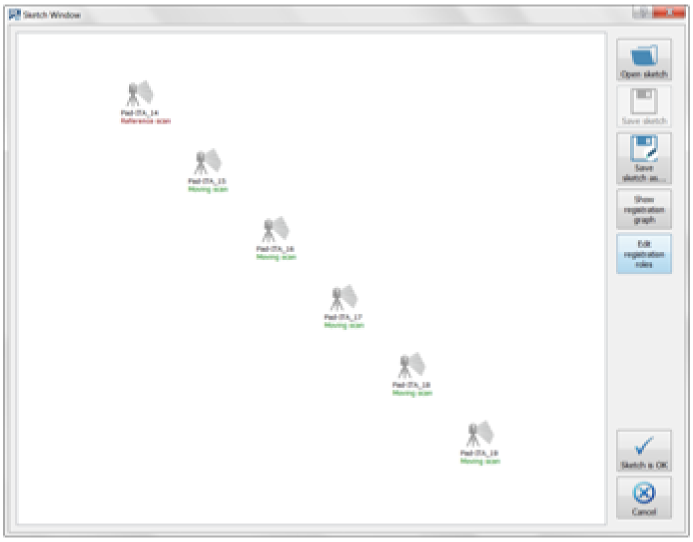
This function automatically registers a set of scans by detecting and matching targets. The user can start the automatic target detection, otherwise can manually add, move and remove targets.

1b. Manual Preregistration (among models)

This function allows you to manually compute a rough alignment between two generic models. The procedure works by finding three couples of corresponding points among the reference and moving models.

A particular case of this command is the registration between two grid point clouds (see Manual Pre-registration among grid point clouds).

Note: in the target-free automatic preregistration process there's a possibility to use a sketch to aid the computation, suggesting the relative position of the scans (see Sketch Window)

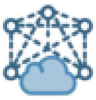


2. Fine registration

2a. ICP Registration (cloud to cloud)

ICP Registration is an algorithm to automatically perform fine registration of a moving point cloud against one or more reference clouds. The moving cloud must be roughly close to the reference cloud.

2b. Bundle Adjustment



This algorithm allows to register many point clouds, distributing evenly the registration error. The user specifies which point clouds are reference clouds (they are locked during the registration) and which are moving; the moving clouds will move and align on the reference clouds and between them during alignment.

At the end of the processes you should see your scans well aligned.

Georeferencing

The **LineUp®** toolbox includes a procedure to geo-reference the imported point clouds, through reference points

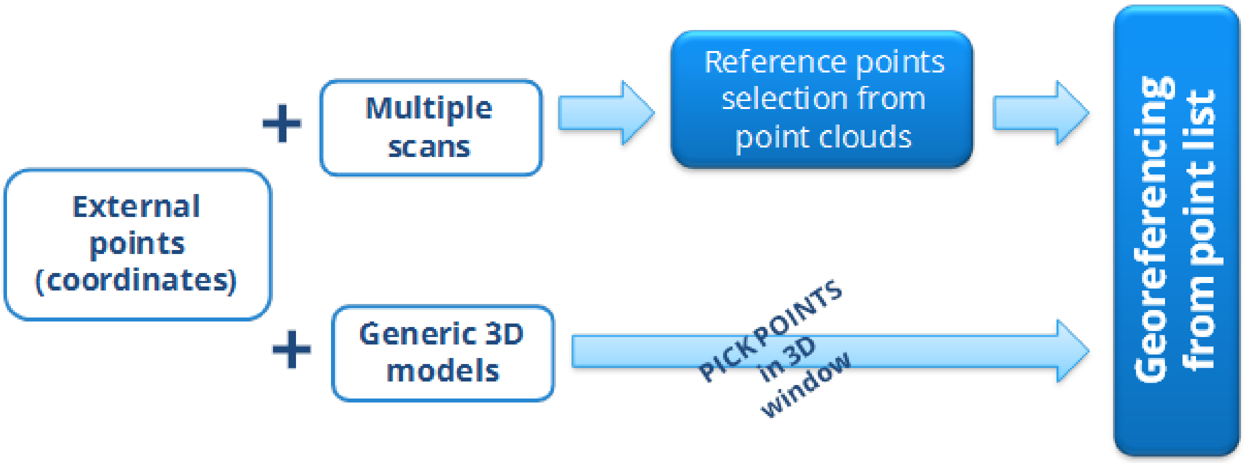
Point clouds georeferencing

Point list registration

To georeference any items or entire projects using two point lists

This process allows you to georeference multiple scans using both targets (or other points) and the centers of the scans (if geo-referred).

You can georeference one or more scans by selecting at least 3 points from different scans, known also in another reference system (for example from topography). In the same way a georeferentiation of generic 3D models (i.e mesh models from BIM) is possible (see 3D Model georeferencing).



These steps will be illustrated in the following:
1. Reference points selection
2. Scans georeferencing
3. 3D Model georeferencing

To find and manage reference points on the point clouds in the project, you can use the commands:

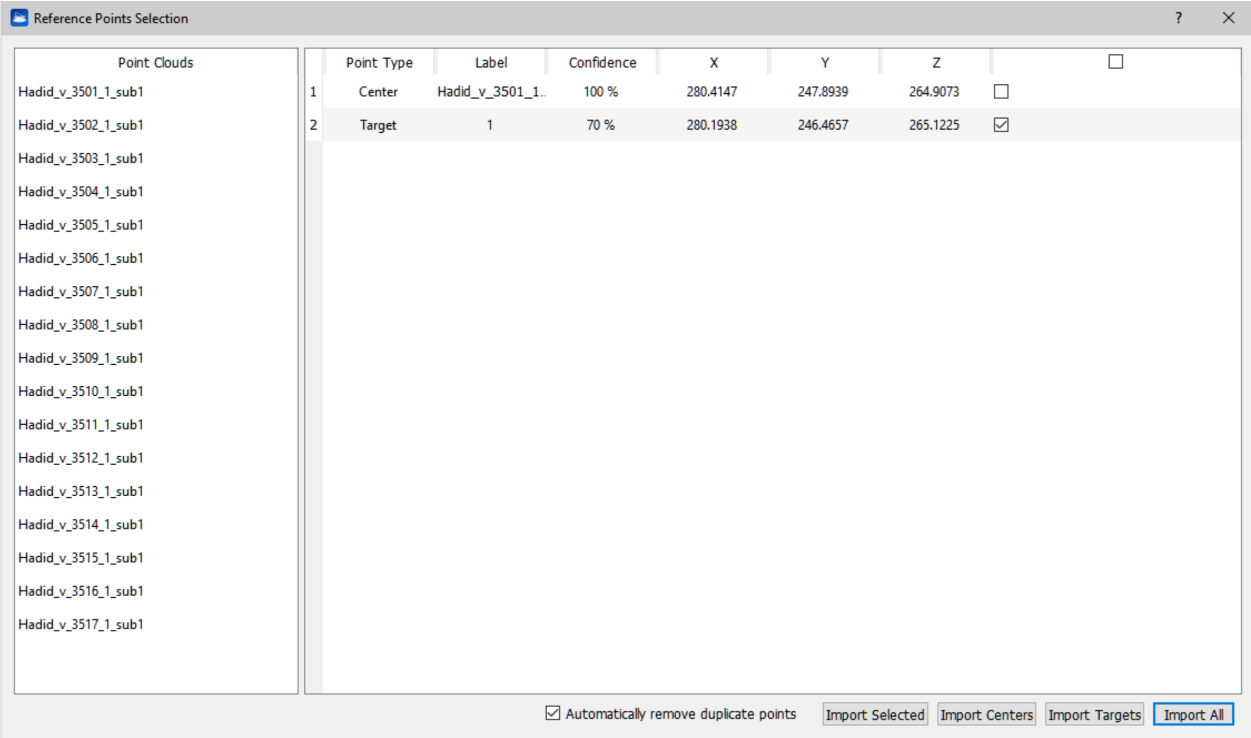

Manage control points
To manually pick points on a generic object in the point cloud and to manage them


Manage circular targets
To automatically find the center of planar circular target or manually pick points on a generic object in the point cloud


Manage spherical targets
To automatically find the sphere center

1. Reference points selection

When you select the *Point clouds georeferencing* command and at least one point cloud (grid or unstructured) the *Reference points Selection* window appears. This dialog allows the selection of multiple reference points from the point clouds target and center points. Target points can be automatically or manually set using the targets editor dialog.

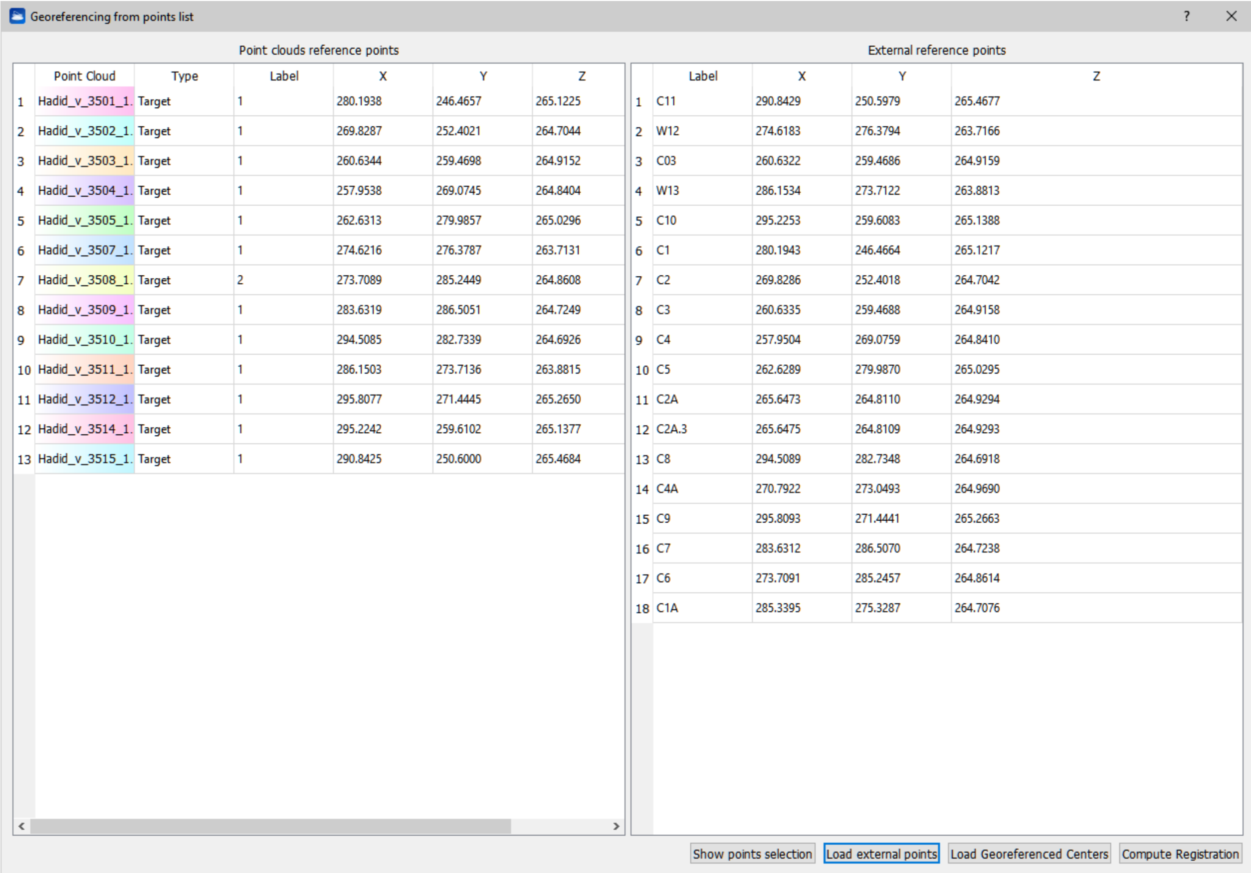


The reference points taken with Manage control points and Manage circular targets procedures will be automatically loaded in the *Reference Points Selection* window.

See all the details in *Target-based pre-registration*.

2. Scans geo-referencing

After the reference points selection, using *Georeferencing from point list* dialog allows you to *Load external points* (or *Georeferenced Centers* of the scans) to geo-reference scans by using previous selected target points.



Georeferencing from points list

Point clouds reference points

	Point Cloud	Type	Label	X	Y	Z
1	Hadid_v_3501_1.	Target	1	280.1938	246.4657	265.1225
2	Hadid_v_3502_1.	Target	1	269.8287	252.4021	264.7044
3	Hadid_v_3503_1.	Target	1	260.6344	259.4698	264.9152
4	Hadid_v_3504_1.	Target	1	257.9538	269.0745	264.8404
5	Hadid_v_3505_1.	Target	1	262.6313	279.9857	265.0296
6	Hadid_v_3507_1.	Target	1	274.6216	276.3787	263.7131
7	Hadid_v_3508_1.	Target	2	273.7089	285.2449	264.8608
8	Hadid_v_3509_1.	Target	1	283.6319	286.5051	264.7249
9	Hadid_v_3510_1.	Target	1	294.5085	282.7339	264.6926
10	Hadid_v_3511_1.	Target	1	286.1503	273.7136	263.8815
11	Hadid_v_3512_1.	Target	1	295.8077	271.4445	265.2650
12	Hadid_v_3514_1.	Target	1	295.2242	259.6102	265.1377
13	Hadid_v_3515_1.	Target	1	290.8425	250.6000	265.4684

External reference points

	Label	X	Y	Z
1	C11	290.8429	250.5979	265.4677
2	W12	274.6183	276.3794	263.7166
3	C03	260.6322	259.4686	264.9159
4	W13	286.1534	273.7122	263.8813
5	C10	295.2253	259.6083	265.1388
6	C1	280.1943	246.4664	265.1217
7	C2	269.8286	252.4018	264.7042
8	C3	260.6335	259.4688	264.9158
9	C4	257.9504	269.0759	264.8410
10	C5	262.6289	279.9870	265.0295
11	C2A	265.6473	264.8110	264.9294
12	C2A.3	265.6475	264.8109	264.9293
13	C8	294.5089	282.7348	264.6918
14	C4A	270.7922	273.0493	264.9690
15	C9	295.8093	271.4441	265.2663
16	C7	283.6312	286.5070	264.7238
17	C6	273.7091	285.2457	264.8614
18	C1A	285.3395	275.3287	264.7076

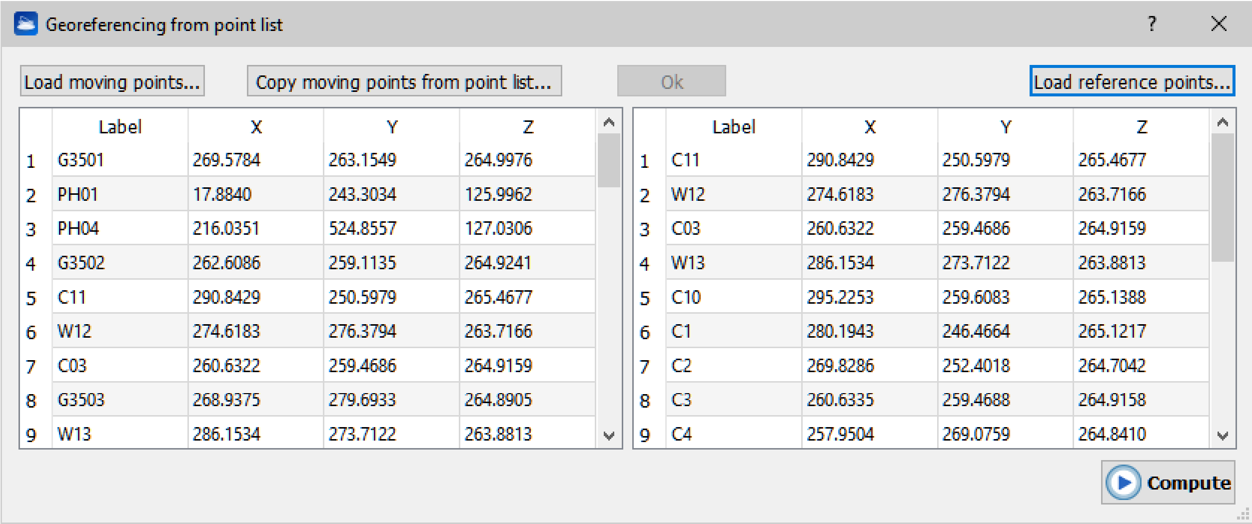
The *Compute Registration* command automatically finds the best matching points from the right and left list and computes the alignment between them.

3. 3D Model georeferencing

Using the *Point list registration* command in Reconstructor® environment, you can register a list of points, named *moving* points, against another list of points that you consider as *reference* (these last points can be geo-referenced). So you can take some points on the model and register it using geo-referenced coordinates of the same points.

Here the process:

- *Moving* points (in the left half of the dialog): you can load a text file with listed points or you can copy the points listed in the project
- *Reference* points: (in the right half of the dialog): you can load the reference points from a text file
- press *Compute* to register the points by coupling them. A dialog appears, asking whether you want to register the points by coupling them according to their labels, or by trying out all the possible pair combinations to find the best. The first option is much faster but it assumes that you are sure about how to match your points. After you have selected either *Match names or Best fit*, you can refine and make use of the results of the registration, in the Registration report dialog.



Note: to start the registration, it is not necessary for the left table to contain as many points as the right table.

It’s suggested to save the scans position before to move them in the 3D space, so it will be easy to recover the previous position in case of a bad displacement or alignment (see also Restore a Pose).

For more details see Georeferencing from point list page.

Points Filtering and Clustering

This category of tools includes functions dedicated to work with point clouds, to enable other processes and further results. Most of these tools work on any point clouds, some of them work only on grid point clouds.
The filtering tools include also the point cloud Pre-process filters: Reconstructor® applies a set of algorithms to the scans which extract information that is needed during further processing of the data.
All the commands can be activate also through the point clouds contextual menu.

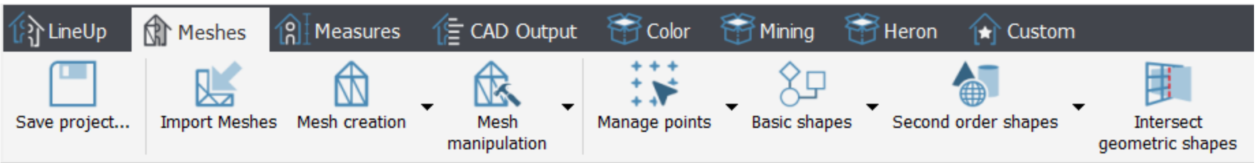
<table>
<tr><td colspan="3">Processes</td><td>On Grid Point Clouds</td><td>On Unstructured Point Clouds</td></tr>
<tr><td rowspan="2">Filtering</td><td rowspan="2">Pre-process clouds</td><td>• Noise Removal (Range & Reflectance Gate, Outlier Removal, Median Filter, Mixed Point Filter, Noise Remover)
• Compute Normals
• Edge Detection (Depth & Orientation Discontinuity)
• Compute Confidence</td><td>✓</td><td></td></tr>
<tr><td>• Noise Removal (Outlier Removal, Noise Remover)
• Compute Normals
• Compute Confidence</td><td></td><td>✓</td></tr>
</table>

	Restore raw data	To undo any operation of preprocessing, deletion and editing that may have been performed on the clouds.	✓	✓
	Restore deleted points	To undelete all the points earlier deleted.	✓	✓
	Edit 2D	A grid point cloud is shown in its 2D representation. Here you can select, delete and undelete points with several functions.	✓	
	Fill holes	To replace any invalid point in the cloud with a value averaged from the point's neighbourhood in the cloud's structure.	✓	
	Hide black points	To invalidate all the points in the cloud that are colored in full black.	✓	✓
	Remove duplicated points	To invalidate any point that has exactly the same coordinates of another point in the cloud.	✓	✓
	Resample	To resample a point cloud, subsampling it.	✓	✓
	Simplify points	To determine the most relevant points from a point of view of shape description, and save them into the new unstructured point cloud. These resulting clouds work as compact representations of the original structured ones.	✓	
	Extract edges	To extract the edges of a grid point cloud, in form of polylines.	✓	
	Level 3D density of clouds	To cluster clouds excluding duplicated or unneeded points. The resulting cloud, however, will not contain all points from the input clouds, but only those needed to guarantee a fixed 3D density of the points.	✓	✓
Clustering	Level 3D density of clouds	To cluster clouds excluding duplicated or unneeded points. The resulting cloud, however, will not contain all points from the input clouds, but only those needed to guarantee a fixed 3D density of the points.	✓	✓
	Make single cloud	To lump together in an unstructured point cloud an arbitrary set of point clouds.	✓	✓

Virtual scan	To resample the scene and generate a new clustered grid point cloud.	✓	✓

See Cloud Processing for details.

Meshes



Manage meshes

A triangle mesh is a 3D model represented by a set of triangles connected by common edges and common vertices. A triangle mesh therefore defines a surface in the 3D space.
In this section an overview of the techniques to create and edit meshes will be presented.

Manage Points and Shapes

In Reconstructor® you can also create, manage and edit some geometric shapes.
In this section an overview of how to create and manage them will be presented.

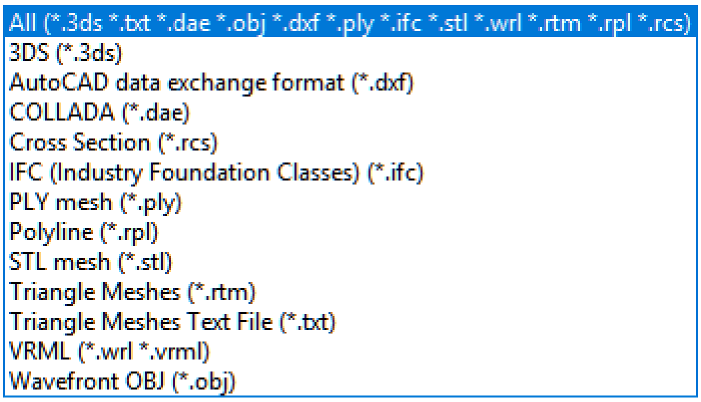
Manage Meshes

Several tools, starting from the mesh import to the mesh creation and manipulation are available.

In the next paragraphs a short overview.

Import Meshes

Reconstructor can import various type of mesh files; below all of them are listed.



Some type of mesh need specific interface to be imported, you'll see it at the import start.

Mesh creation

Reconstructor® provides 4 typologies of mesh techniques. Different number and type of point clouds can be used as a basis to create a mesh, depending on the meshing technique:

	On Grid Point Clouds	*On Unstructured Point Clouds*	*Suggested for...*

Multiresolution Mesh Fast meshing technique that give back light meshes that may have holes in some situations.	✓ (one or more)		To obtain a well defined and fast mesh from a single structured point cloud, with a good quality/computational time ratio
Mesh from predefined view Relatively slow meshing technique that gives back convex meshes without holes. It's a view dependent, high defined mesh (each point is a vertex)	✓ (one)	✓ (one – single or clustered point cloud)	Useful for façades (using orthocamera) and tunneling (using cylindrical camera)
3D Mesh Approximative 3D meshing not view-dependent and taking as constraints the points' positions and orientations (normals)	✓ (one)	✓ (one – single or clustered point cloud)	Useful for convex surfaces.
Topographic Mesh Fast meshing algorithm designed for DTM models. It gives back a watertight, light, smoothed mesh useful for isolines and volumes calculation.	✓ (one or more)	✓ (one or more)	Useful for land survey and mining

Mesh manipulation

Mesh Editor

An editing environment to perform advanced operations such as hole-filling, borders detection, editing triangles and vertexes, smoothing surfaces, decimating, crests and toes extraction.

Merge meshes into one

This dialog allows to lump together in a single mesh an arbitrary set of triangle meshes

Convert mesh to point cloud

To create an unstructured point cloud from mesh.

Get mesh borders as polyline

To create a new polyline containing the mesh's borders and add it to the project

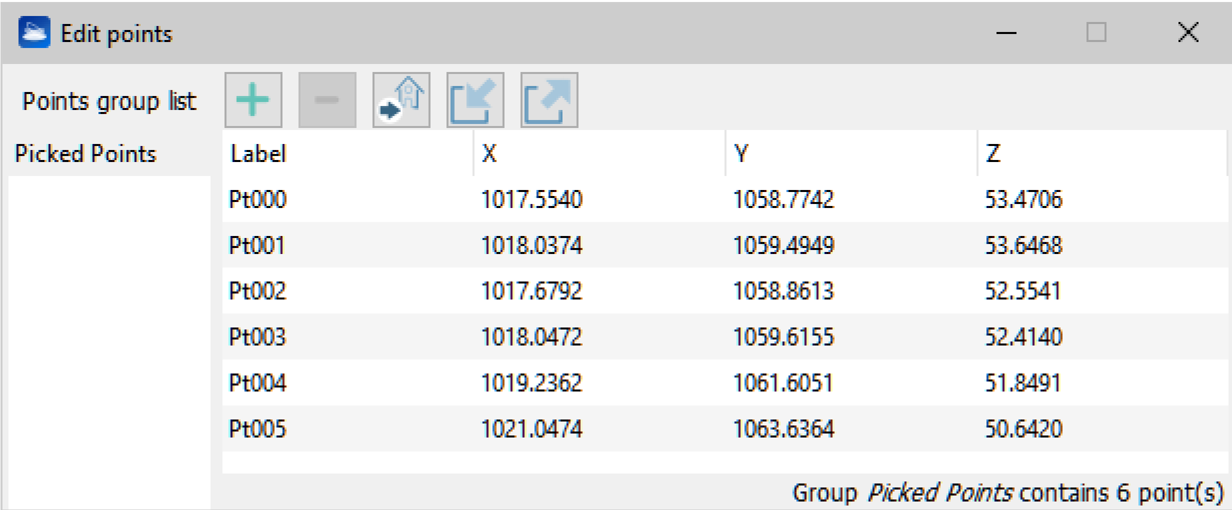
Mesh selection from current view point

To cut a portion of a mesh using 2D video selection tools on the current view.

Reconstructor® provides many other functions for editing meshes.

Manage points and shapes

Manage points



By Alt + Left Double Mouse clicking you can pick a point on a model and save it in the "Picked Point" list.
It is possible to import a point list from a text file: *File menu* -> *Import a list of points...*
Collected points can be also used to fit primitives, draw polylines or convert point to targets.

Manage Points

- Edit point list
- Export point list
- Polyline from point list
- Fit shape to point list

Primitives (planes, cones, cylinders, spheres, circles) can be created by a robust fitting algorithms. An accurate fitting reports is also provided.

Basic Shapes

- Create/edit plane
- Create/edit segment
- Create/edit polyline

Second Order Shapes

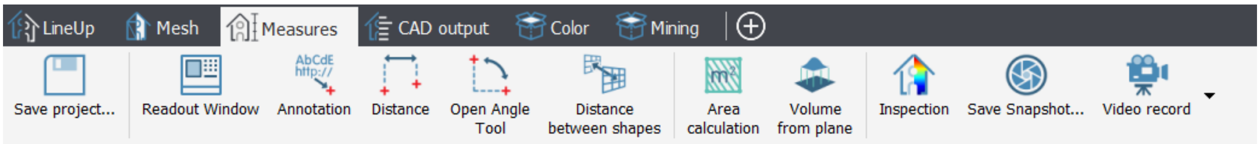
- Create/edit circle
- Create/edit sphere
- Create/edit cylinder
- Create/edit cone

Shapes Interactions

Minimum distance between fitting primitives can be quickly calculated; bridge clearance, building heights, poles/pillars inter-distances, tank diameters can be easily extracted.
Fitting primitives can be intersected providing precise polylines.

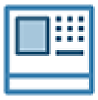
- Distance between shapes
- Intersect geometric shapes

Measures



Reconstructor® provides different tools to take measurements in the 3D space, as annotation, distances, angles, area and volume computations.

In Reconstructor® some basic measuring features are available:

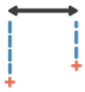


Readout window

The readout window allows you to see the 3D coordinates of the points you are hovering the mouse on in the 3D rendering window.

Annotation

This function allows you to create annotation of 3D points of models in your project.



Distance

This button allows you to measure a distance between any two points in the 3D scene.

Open Angle Tool

This button enables you to measure and angle between three points in your 3D scene.

Area and Volume

To take measures as:

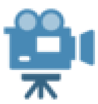
- Area
- Volume from plane
- Compute cut&fill volume

Inspection

It is a procedure to compare two different shapes and to measure their differences.

Save snapshot

This function allows you to capture a snapshot of the main viewport with its rendered models. You can save the captured snapshot in a variety of image formats, including Bitmap, Jpeg and PNG.

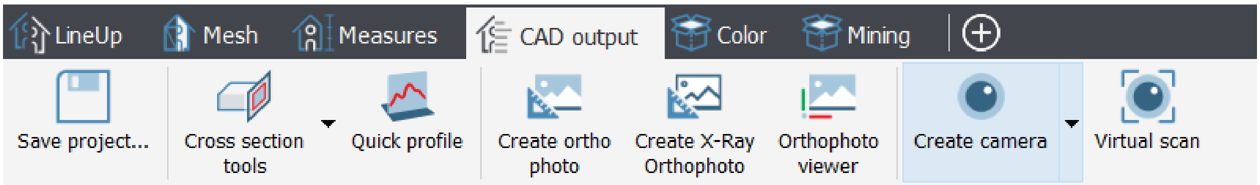


Video record

To acquire and save video navigating in the 3D window.

More details at Measures & Notes.

CAD Outputs

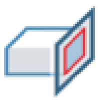


CAD Output

Cross sections, Quick profiles, Orthophotos and X-Ray Orthophotos can be created, edited, and used to take measurements in the 2D space.
All these outputs can be extracted from point clouds (both grid and unstructured) and meshes.

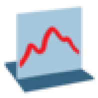
Points and polylines can be picked and created in the 3D space, but you can constraint them on a plane (to create a 2D entity) by using Drawing tools.

All these items can be exported into CAD software.



Cross Section Tools
It contains commands and features to create cross sections and isolines of point clouds (grid or unstructured) and meshes, starting from cutting planes and other constraints:

- Create/edit plane
- Cross section
- Section from plane



Quick profile
Creates ***vertical cross sections*** of models by simply picking two points in the 3D window you can access the relative command in the top toolbars.

The following commands are used to create, manage and visualize precise, high-resolution elevations and plans of your models:

Create Orthophoto
Creates an Orthophoto and defines its position, size and field of view based on different parameters and methods.

Create X-Ray Orthophoto
Creates an X-Ray Orthophoto from a set of selected point clouds. Main edges and borders are emphasized and displayed in the final Ortho view.

Orthophoto viewer
Opens an orthophoto in the viewer to measure distances, angles and areas.
From the viewer, the orthographic image can be exported in AutoCAD.

The following tools are used to extract cameras and run virtual scans by using them.

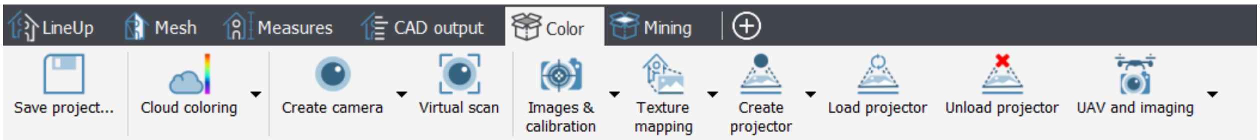
Create Camera
Creates four types of camera (perspective, ortho, cylindrical and spherical).

Place here survey point
Creates a perspective camera placed in the current view point.

Virtual scan

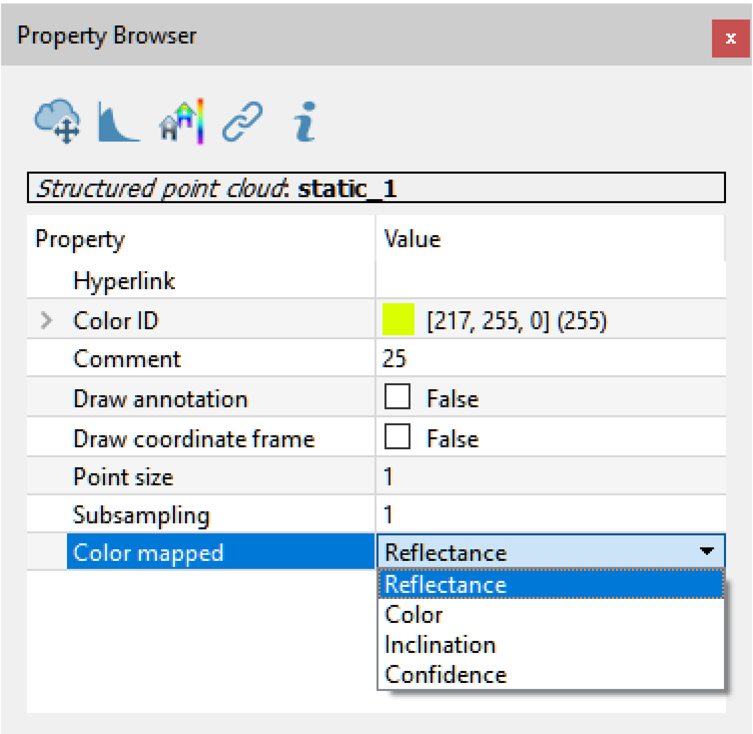
Computes a Virtual Scan of the models in the 3D scene from a selected camera. The virtual scan can be saved as a 2D image, or as a grid point cloud, and exported to AutoCAD as a scaled Orthophoto.

Color



Reconstructor® provides several coloring tools to effectively manage the colorization of point clouds and meshes.
These tools can be classified according to the objects you want to color and the procedures you need to apply:

- Some color information is item intrinsic of point clouds and saved as a color layer (see Cloud coloring).
- A mapping (on point clouds and meshes) of an image from external cameras is possible (see Camera calibration, projectors and colorization).

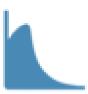


To look at point clouds' color layers, select the point cloud from the project window and set the *Color Mapped* option in the Property Browser.

Open the Photo & Color detailed page.

Cloud coloring

The following commands help the user to manage the color layers of point clouds.



Histogram

To optimize the point clouds color's contrast by histogram stretching. Only single channel layer can be used (reflectance, gray scale layers...).
This command is available in the *Property browser* dialog.

Colors mapping
To manage an artificial colorization for a given *color layer* of a point cloud. Color information can be scalar (e.g. reflectance, range, confidence) or vectorial (e.g. inclination). It's also possible to add layers from external images.

Color with altitudes
To add to the selected point cloud(s) an extra color layer, representing the altitude of the points with respect to one of the tree axes of the current UCS.

Inclination from plane
To add to the selected point cloud(s) an extra color layer, representing the inclination of the points' normals with respect to a given plane that exists in the project.

Use it to classify points for a further vegetation removal.

Color with range
To add a layer coloring the distance from the relative observation point.

Color with ambient light
To add to the selected point cloud(s) an extra color layer,

Visualization of outdoor large mapped areas can be enhanced with this layer.

Compute 3D reflectance
To add to the selected point cloud(s) a special color layer, named *3D Reflectance*, that mixes LiDAR intensity and point normals (*inclination*) to enhance both geometrical and surface characteristics.

Add color layer from image
To apply an image on the point cloud (e.g. a 2Dview after a color editing).

Merge or split layers
To combine RGB and scalar layers to obtain a customized new layer of color for point clouds.

See Photo & Color for more details.

Camera calibration, projectors and colorization

Among the coloring tools, there are some dedicated to colors related to photos.

Images & calibration
To calibrate images on point clouds and meshes and determine the intrinsic parameters of a camera (if necessary):

- Camera calibration on a model
- Camera calibration using 3D points
- Calibrate camera intrinsic

Texture mapping
To color point clouds and meshes by using external images (and also extracted panoramic cameras from perspective picture in Reconstructor®)

- Color clouds with photos

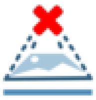
- Create panorama
- Create texture map

Create projector

To create a Perspective, Orthographic, Spherical, Cylindrical projector or from a calibration (by importing a Reconstructor® camera calibration file) using a valid image.

Load projector

To load (light on) the image on the 3D model.



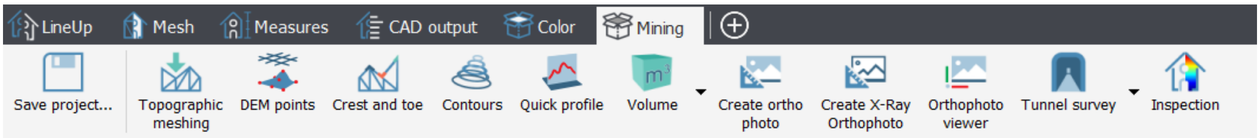
Unload projector

To unload the image from the 3D model.

UAV and imaging

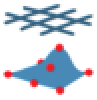
These tool is used to Import GeoTIFF as an orthographic camera.

Mining



In this toolbar there are many different tools to extract information and geometrically analyze the 3D models in mining and tunneling environments.

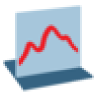
Topographic meshing
To transform a mesh or an unstructured point cloud into a structured one from provided cameras.



DEM points
To extract simplified models of your models (grid or unstructured point clouds and meshes).

Crest and toe
To extract discontinuities from mesh models.

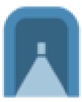
Contours
To create *isolines (contour lines)* of your models (grid or unstructured point clouds and meshes)



Quick profile
To create *vertical cross sections* of your models (grid or unstructured point clouds and meshes) by simply picking two points in the 3D window.

Volume
To compute volumes between a surface and one or more reference surfaces (mesh or planes)

Orthophotos
To create orthophotos and X-ray orthophotos and visualize them.



Tunnel survey
To extract cross sections and cylindrical projection along a trajectory.

Inspection
To compare two different shapes and to measure their differences.

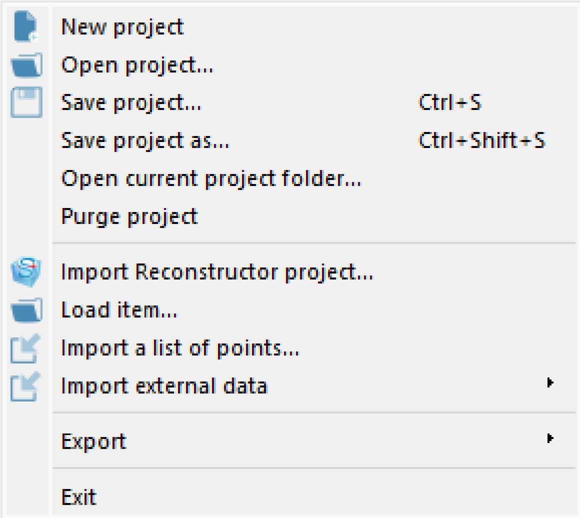
Export

This menu contains a number of options to export Reconstructor®'s entities in various formats, to be imported in third-party software or CAD tools for further elaboration.

- Export model as
- Export polyline as
- Export cross section as
- Export UCS to AutoCAD
- Export annotations
- Export pose
- Export to Autodesk ReCap®
- Export point list

See Export for further details.

File menu



This menu is mainly organised in three main parts:

PROJECT MANAGEMENT	• **New project**: every project is organized in a folder that groups data by model structure and object type, thus the following dialog allows to create and select a folder for the project • **Open project**: opens an existing project (*.recprj format) • **Save project [Ctrl+S]:** saves the current project • **Save project as [Ctrl+Shift+S]**: saves a copy of the project with a different name • **Open current project folder [1]:** opens the folder including the current Reconstructor® project • **Purge project**: can empty the "Trash" project's folder, erase the temporary files, the orphaned item files (badly connected with the project) and backup data.

IMPORT	• **Import Reconstructor Project**
	• **Load item**
	• **Import a list of points**: imports a .txt file as a list of points inside the project
	• **Import external data**: imports data from third party formats
	    o Grid Point Clouds
	    o Triangle Meshes
	    o Unstructured Point Clouds
	    o Polylines/trajectories
	    o CAD Models
	    o F6 handheld scans (.mvx)

EXPORT	• Export model as • Export polyline as • Export cross section as • Export UCS to AutoCAD • Export annotations • Export pose • Export to Autodesk ReCap®

	• Export point list

Exit: quits Reconstructor®. If you have unsaved changes to your projects, Reconstructor® asks you if you what to save the changes or not before exiting.

[1] Each project has its own folder called withthe same name. Reconstructor® projects contains at least some of the following subfolders:
Documents / Exports / Grids / Images / Imports / Meshes / Polylines / Results / Settings / Trash / Unstructs.
These folders are automatically created when the project gets populated.

Items removed from the Reconstructor ® project are moved from their initial location to *Trash* folder.
To recover space on your hard drive, it is really advisable to periodically empty this folder if the contents are no longer useful.

The **Purge project** data tool can help you to save space on disk, detecting and deleting the useless huge data.

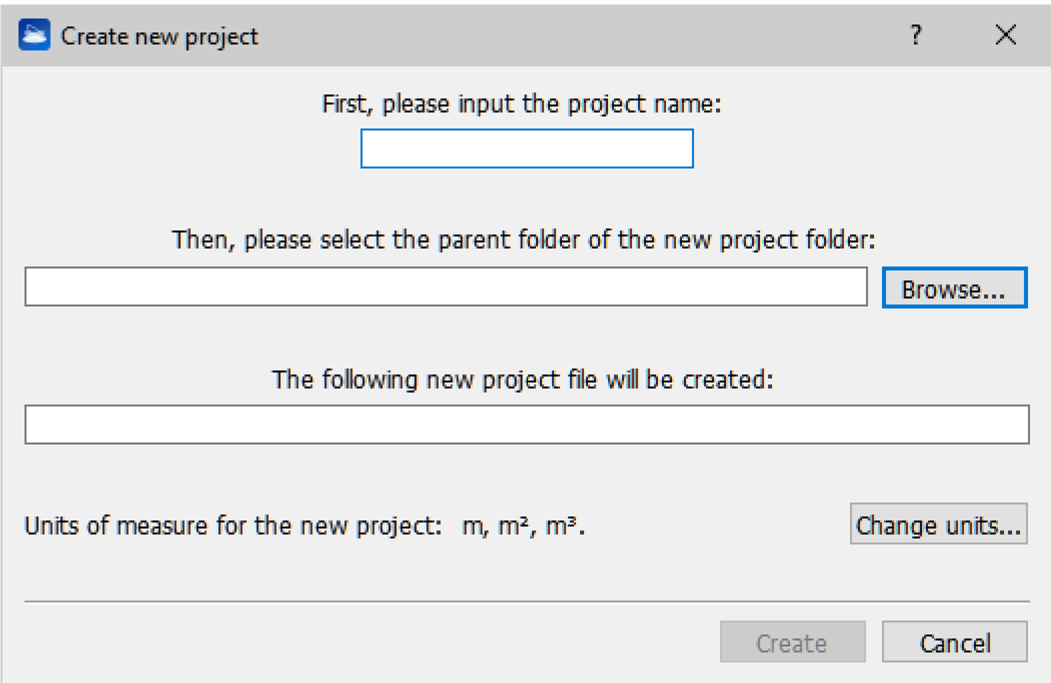
Project Management

In this section you can learn how to efficiently manage your Reconstructor® project.

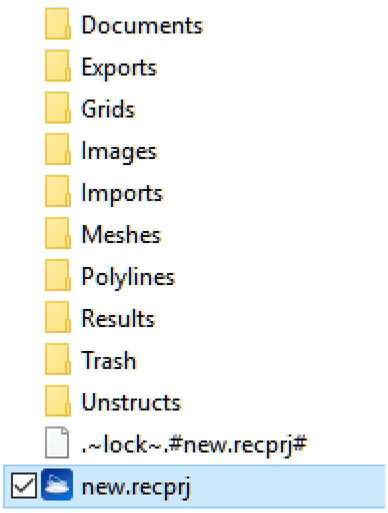
New project

Run Reconstructor®, a splash screen is shown and this interface appears:

Press *New project* button and fill in the gap:



Each project has its own folder called with the project name. Reconstructor® projects contains at least some of the following subfolders:



These folders are automatically created when the project gets populated.
The option to create a new project appears also in Reconstructor® file menu, and has the same functionality.

Open project

This function opens a Reconstructor® project (*.recprj, *.r2s). Before opening a new project, Reconstructor® asks whether you want to save the current one (if you left unsaved changes).
The opening of the same project by two different instances of Reconstructor is not allowed; a lock file is created in the project folder.

Compatibility across different Reconstructor® versions

Reconstructor® projects are backwards compatible. For Reconstructor® to be able to open a project, the project must have been created/saved with a Reconstructor® version with the two most important version numbers are smaller or equal. For example, Reconstructor® 4.0 can open projects saved with versions 3.x; however it cannot open projects saved with Reconstructor® 4.1. However, Reconstructor® 4.0.0 can open projects saved with Reconstructor® 4.0.1.

Persistence of camera pose

When opening a project, Reconstructor® loads all the project items in the project window, and sets the 3D scene's current viewpoint to the position and orientation that it had when the project was saved.

Moving/copying a project

When transferring or copying a project, the whole project folder with all its subfolders must be copied, and not only the project file (*.recprj).

Save project

This function saves the current project, safely storing all the project items, their load/unload state, their properties, the current 3D camera pose, etc.
It is advised to save the project frequently, especially before long processing operations.

Save project as

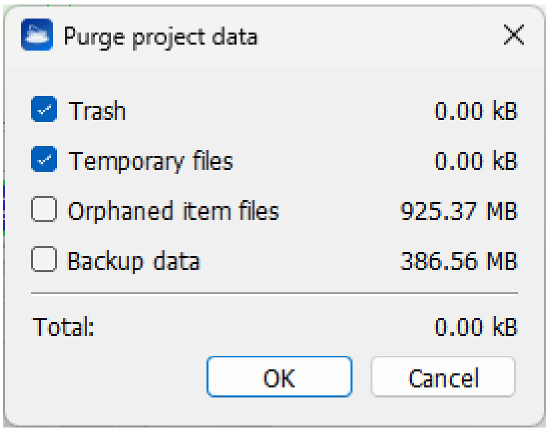
This function saves only the project file (*.recprj) with another name. It is useful for projects with many items, it is possible to have a project with the clouds, and another with the clouds and more accessory items like camera, planes or polylines.
It is also useful to create a copy of the project, by using this function to create another project file in another directory, and then copy all the source project subfolders in the destination project folder.

Open current project folder

This function allows you to quickly open the folder where the current Reconstructor® project is stored.

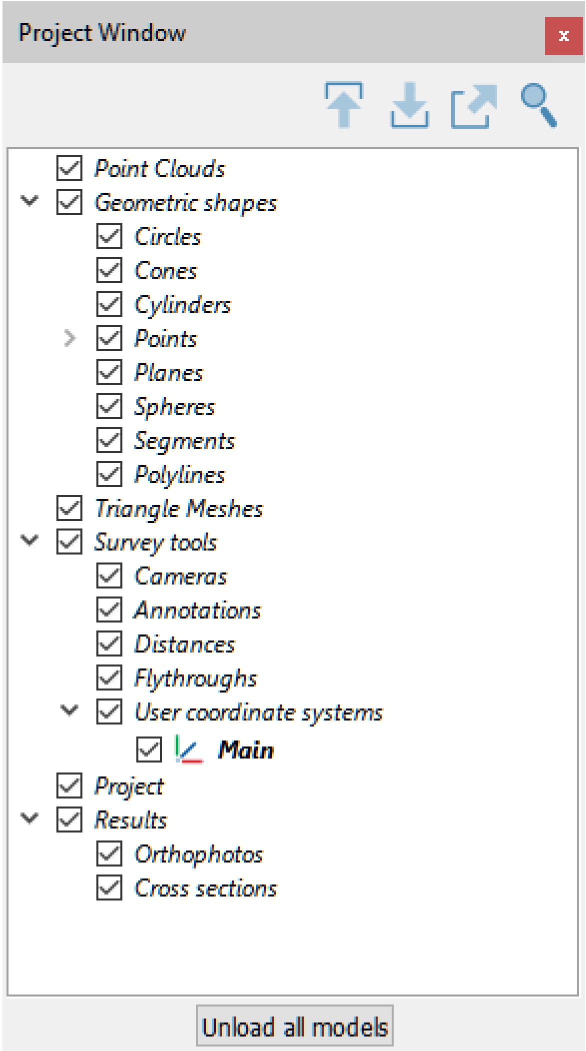
Purge project

This tool can help you to save space on disk, detecting and deleting the useless huge data, as "Trash" project's folder contents, temporary files, orphaned item files (badly connected with the project) and backup data.



Project tree structure

All the Reconstructor® items are subdivided by groups, down here a view of an empty project.



You can cut&paste or drag&drop items into groups.

Import

To quickly import data, just drag and drop the files (or folders) you want to import anywhere in the 3D scene. A wizard will appear to ask you for import options, depending on kind of data.

a. If you're importing a point cloud the **LineUp® Wizard** will be opened.
b. If you're importing a different file format (as meshes, polylines, etc...) a wizard window will appear to guide you through importing steps.

All imported models' coordinates will be interpreted as referred to the current UCS (User Coordinate System).

See Getting Started > Import to learn the fast way to import your data.

Below the main families to importable data:

- **Import Rec Project**
- **Load Item**
- **Import a list of points**
- **Import external data** (file formats)

Import Reconstructor Project

These function imports items from another Reconstructor® project (*.recprj, *.r2s formats are available). The items data can be *copied* or *moved* from the project to be imported.

Pay attention to the current UCS of the project you are importing into.

Load item

This function allows you to import in the current project a pre-existing item saved in Reconstructor internal format. The item is not duplicated, copied or moved from its location. It is just listed as belonging to the project.

With this function, you can load any of the following item types:

- Unstructured point clouds (*.rup)
- Grid point clouds (*.rgp)
- Triangle meshes (*.rtm)
- Cross sections (*.rcs)
- Polylines (*.rpl)
- Trajectories (*.rtr)
- XML item file (*.xml)

These are all Reconstructor® internal formats for items; they are stored in sub-folders of Reconstructor® main project folder.

Do not move or delete these files from their own folders, the project may be corrupted

by this action (items are displayed in grey, in the project tree).

Import a list of points

This function imports a list of points (in *.txt, *.csv, *.tsv formats) with labels and 3D coordinates.

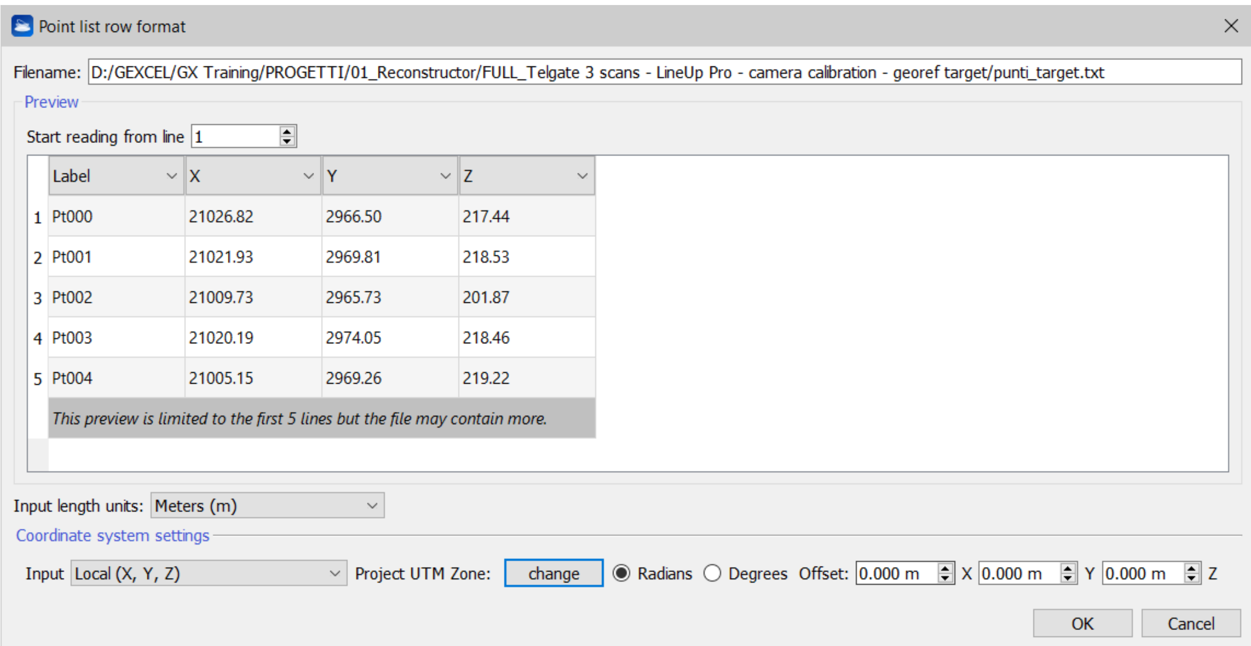
Three formats are available:

- Comma Separated Value (*.csv)
  Comma "," is used as separator.
  ```
  Pt000,-13.9153,-15.7213,11.9317
  Pt001,-36.6361,15.7372,6.3014
  Pt002,-15.1782,6.6534,8.5348
  ```

- Tab Separated Value (*.tsv)
  Tab "*Space*" is used as separator.
  ```
  Pt000   -13.9153   -15.7213   11.9317
  Pt001   -36.6361   15.7372    6.3014
  Pt002   -15.1782   6.6534     8.5348
  ```

- Text (*.txt)
  Tab "*Space*" is used as separator.
  ```
  Pt000   -13.9153   -15.7213   11.9317
  Pt001   -36.6361   15.7372    6.3014
  Pt002   -15.1782   6.6534     8.5348
  ```

Parse point list dialog



This dialog implements a general-purpose tool for parsing a text file (*.txt) with a list of points.

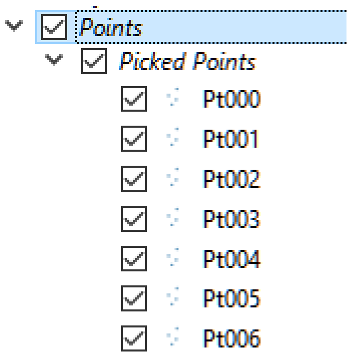
The assumption is that each row of the text file contains one point. The dialog allows you to define how you want to interpret the columns of the text file. In the example above, the first column contains the label of the point, the three next columns contain the 3D coordinates of the point, and

columns five and six contain the X and Y coordinates of the point in the range image (or in any image).
You can customize the role of each column of the file by clicking on a column and dragging & dropping it

In some cases it is useful to invert X and Y columns to adapt the importing reference system to the project one, when regional conventions are different, by dragging&dropping the column (not selecting the title row) you want to move in the right order.

The preview of the list of ponts is limited to the first 5 lines to but the file may contain more.

At the end of the import process, the points are merged in the *Points* group of the Project Window.



Import external data (file formats)

To import data from third party formats. The formats are grouped by the way adjacency structure is preserved, so first select the structure of the data that must be imported. The following dialog allows to browse and filter the directories by file type. The "Files of type" combo box lists the available import filters. Since the format filters are based on a plug-in system, this list is extensible and depends on your software distribution. After the model structure is chosen, a dialog asks you which files would you like to import. Please filter the files by the desired format type. Multi selection is possible with shift and control keys.

Import Grid Point Cloud... Imports grid point cloud from disk. Scan data, organized in regular row and columns, from third parties laser scanner are supported	Comma Separated Values (*.csv) DEM ASCII header+matrix (*.asc) E57 grid point cloud (*.e57) [1] Generic binary traster (*.*) Grid Point Cloud Text File (*.txt) PTX (*.ptx) [1] Stonex multiple scans (*.x3m) Stonex point cloud (*.x3s)
Import Triangle mesh...	3DS (*.3ds) AutoCAD data exchange format (*.dxf) COLLADA (*.dae) PLY mesh (*.ply) STL mesh (*.stl) Triangle Meshes Text File (*.txt)

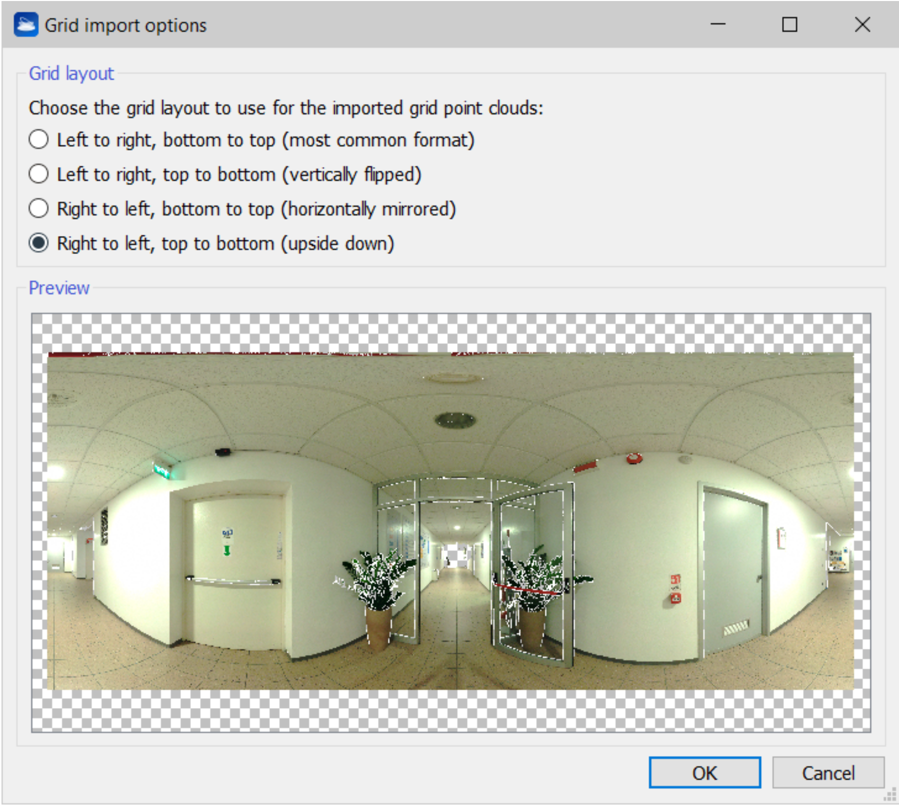
	VRML (*.wrl, *.vrml) Wavefront OBJ (*.obj)
Import Unstructured Point Cloud...	E57 unstructured point cloud (*.e57) LAS format (*.las) LAZ format (*.laz) PLY point cloud (*.ply) Unstructured Point Cloud Text File (*.txt) PTS (*.pts)
Import polyline / trajectory...	AutoCAD data exchange format (*.dxf) DXF trajectory (*.dxf) Polyline from point list (*.txt) Trajectory from point list (*.txt)
Import CAD Models...	AutoCAD data exchange format (*.dxf) STL mesh (*.stl) VRML (*.wrl, *.vrml) IFC (Industrial Foundation Classes) .ifc
Import F6 handheld scans	F6 handheld scans (.mvx)

[1] *When importing .e57, .ptg and .ptx structured point clouds formats, you need to choose the grid layout, in order to correctly import the structure and the 2D image.*

Different options are available:

- *Left to right, bottom to top*
  This is the most common format and it is suggested to use it for most of the cases.
- *Left to right, top to bottom (vertically flipped), Right to left, bottom to top (Horizontally mirrored), Right to left, top to bottom (upside down)*
  Use one of these options only when the data is not standard and the 2D image is upside down or horizontally flipped.

A preview helps you to easily find the correct option for your data.



Import CAD models

This function allows you to import in Reconstructor® models from CAD softwares, in order to do structural analyses or to compare the models with the point clouds.

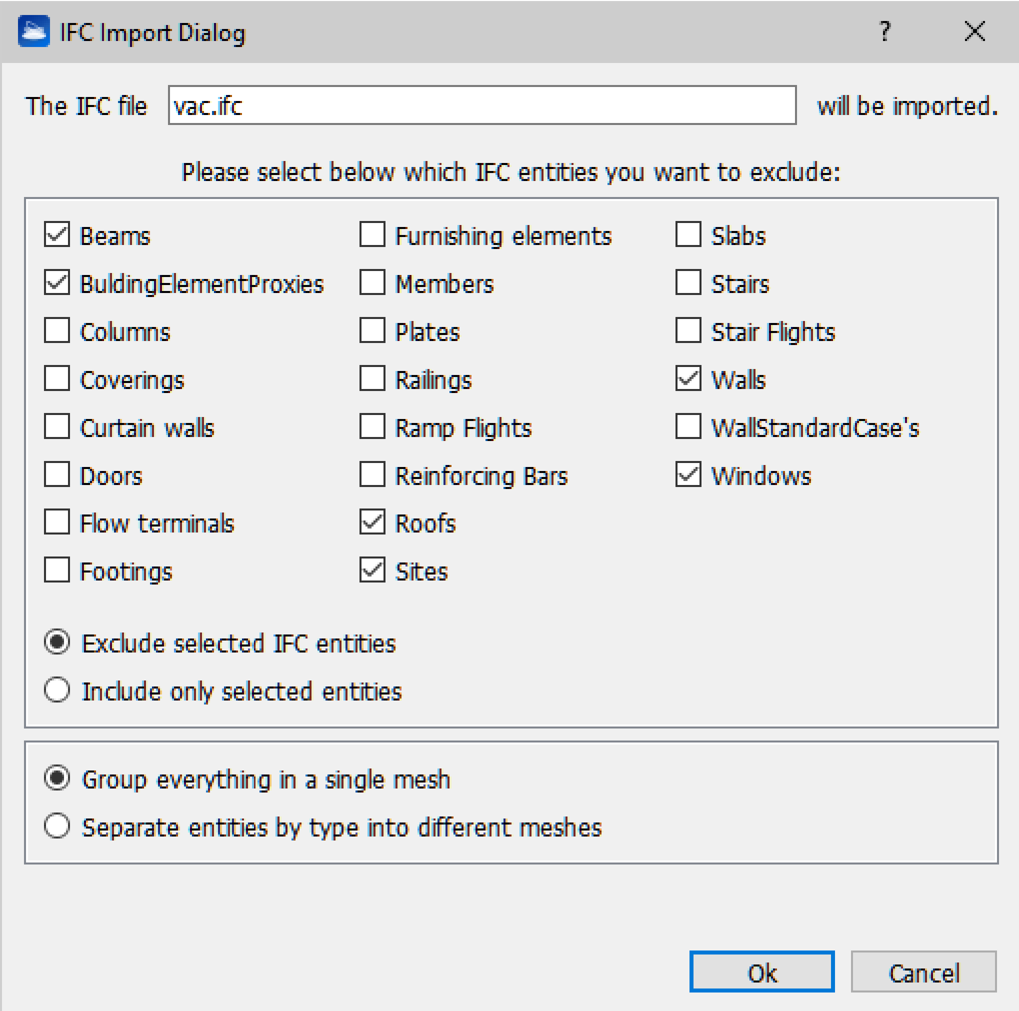
List of supported CAD formats

You can import CAD models in any of the following formats:

AutoCAD data exchange format(* .dxf)	You can import meshes written in DXF format, Reconstructor® will also read their colors and textures.
IFC (Industrial Foundation Classes) (*.ifc)	You can import models e.g. from Autodesk Revit via the IFC format. Reconstructor® preserves the model colors (except for the transparency)
STL Mesh (*.stl)	When importing STL files, Reconstructor® asks you the unit of measure in which the STL vertices are written. Supported units of measure are: meters, centimeters, millimeters, inches, feet, yards If the STL file is binary, and if the string “COLOR=” is present in the first 80 bytes, then Reconstructor® tries also to import the colors of the STL solid. If not, no color is imported. The standard STL format does not support colors neither textures. This is only one of the non-standard formats to write color into STL files

VRML (*.wrl, .vrml)	When you import vrml models as meshes, Reconstructor® will maintain their colors and textures. Reconstructor® will also ask you for a scale factor

IFC Import Dialog



Interoperability with Autodesk Revit®

This import tool allows you to import your BIM project realized with Autodesk Revit® inside Reconstructor®. The BIM entities are converted into meshes and can be compared with point clouds via the inspection functionality, to get accurate deformation and structural analyses.
From Revit®, just export your project in the IFC format, and then simply drag&drop your IFC file inside Reconstructor®. The above dialog opens up.

Exclude unneeded entities

BIM projects can be very large and detailed, including furniture elements, railings, windows, etc. For the sake of inspection, you may want to exclude some project entities because, for example, they are simply not yet in place in reality. To exclude some entity types, just mark them among the check buttons that represent the most commonly used IFC entity types. In the screenshot above, for example, all furnishing elements, railings, ramp flights, sites and “BuildingElementProxy” 's will be excluded from the import procedure. You can also specify which entity types to *include* , by selecting the *Include only selected entities* radio button. In this case, you have to check at least one entity type to proceed importing.

Entities grouping

In the bottom box of the dialog, you have two grouping options: *Group everything in a single mesh* and *Separate entities by type into different meshes.* The first option will merge all your IFC entities in a single mesh, resulting in a faster inspection procedure. The second option will split your IFC entities among many meshes, one by entity type. If you choose the second option, Reconstructor® will create many meshes and put them under the same parent group in the project. The meshes will be called with the following convention: < *IFC file basename*>_<*IFC entity type*>. For example: “MyBuilding_IfcWall”, “MyBuilding_IfcRoof”, “MyBuilding_IfcWindow”, etc.

Export

This menu contains a number of options to export Reconstructor® entities in various formats, to be imported in third-party software or CAD tools for further elaboration.

- Export model as
- Export polyline as
- Export cross section as
- Export UCS to AutoCAD
- Export annotations
- Export pose
- Export to Autodesk ReCap®
- Export point list

Note that when exporting models or other entities, the coordinates are exported as they appear in the current UCS, if not required by an export settings window.

Export Model as...

This function allows to export a model (models are all point clouds and triangle meshes) in a third-party format or in an intermediate format, for further processing in external software tools.

Export workflow

If you want to export a given list of models (clouds or meshes) to the same third-party format, after selecting this function you are presented with a dialog where you can set format-specific options. Below you find a table with the supported options by format, and with the known limitations of each format. You can choose whether to specify different export options for each model, or to adopt the same export options for all models. In the latter case, select *Do not ask again* in the subsequent save filename dialog.

Format limitations when exporting global coordinates

Generally speaking, when exporting a model to an external format, Reconstructor® tries to store as much information as the format allows. Therefore, models are always exported in global coordinates, meaning the coordinates that the model has in the current UCS. However, some format do not allow big coordinates for the models' points. Such formats are STL meshes and PLY meshes. Because of intrinsic limitations of these formats, models exported in these formats are saved in local coordinates, losing their global position in Reconstructor project.

Export formats options and limitations

Grid Point Clouds

Export format	**Supported options**	**Limitations**
E57 grid point cloud (*.e57) E57 unstructured point cloud (*.e57)	Choosing local or UCS coordinates. Choosing whether to export the reflectance, also in a way compatible with Autodesk ReCap. Choosing whether to export a	

	color layer and which one. See more details at **Export E57**	
Generic binary raster (*.*)	Choosing which scalar layer to export.	
Grid Point Cloud (*.rgp)		
Grid Point Cloud Text File (*.txt) Grid Point Cloud CSV File (*.csv) Grid Point Cloud TSV File (*.tsv) Unstructured Point Cloud Text File (*.txt) Unstructured Point Cloud CSV File (*.csv) Unstructured Point Cloud TSV File (*.tsv)	Choosing object or UCS coordinates. Setting a subsampling factor for the exported cloud. Choosing the saved number of decimals. Choosing whether to export a color layer and which one. Skipping the depth discontinuities or not. Skipping the orientation discontinuities or not.	
LAS/LAZ format point cloud (*.las, .laz) [unstructured]	Choosing local or UCS coordinates. Choosing whether to export the reflectance, also in a way compatible with Autodesk ReCap. Choosing whether to export a color layer and which one.	
PLY point cloud (*.ply) [unstructured]	Selecting whether to export local or UCS coordinates. Selecting whether to export a color layer and which one. Selecting whether to export point normals or not. Selecting the storage mode among ascii, binary little endian, binary big endian. Selecting the coordinate precision between float and double.	If the model to be exported has too big UCS coordinates (more than 100 km), Reconstructor will not export it in UCS coordinates, because of PLY format limitations.
PTX point cloud (*.ptx) [structured]	Choosing a subsampling factor. Choosing which color layer to export. Choosing whether to skip depth discontinuities. Choosing whether to skip orientation discontinuities. Choosing whether to export a PNG image of the selected color.	

PTS point cloud (*.pts)	Choosing object or UCS coordinates. Setting a subsampling factor for the exported cloud. Choosing the saved number of decimals. Choosing whether to export a color layer and which one. Skipping the depth discontinuities or not. Skipping the orientation discontinuities or not.	

Unstructured Point Clouds

All the exported point clouds will be in an unstructured format.

Export format	Supported options	Limitations
E57 unstructured point cloud (*.e57)	Choosing local or global coordinates. Choosing whether to export the reflectance, also in a way compatible with Autodesk ReCap. Choosing whether to export a color layer and which one.	
LAS/LAZ format point cloud (*.las, .laz) [unstructured]	Choosing local or UCS coordinates. Choosing whether to export the reflectance, also in a way compatible with Autodesk ReCap. Choosing whether to export a color and which one.	
PLY point cloud (*.ply)	Selecting whether to export local or UCS coordinates. Selecting whether to export a color layer and which one. Selecting whether to export point normals or not. Selecting the storage mode among ascii, binary little endian, binary big endian. Selecting the coordinate precision between float and double.	If the model to be exported has too big UCS coordinates (more than 100 km), Reconstructor will not export it in UCS coordinates, because of PLY format limitations.
Unstructured Point Cloud (*.rup)		
Unstructured Point Cloud Text File (*.txt)	Choosing object or UCS coordinates.	

Unstructured Point Cloud CSV File (*.csv) Unstructured Point Cloud TSV File (*.tsv)	Setting a subsampling factor for the exported cloud. Choosing the saved number of decimals. Choosing whether to export a color layer and which one. Skipping the depth discontinuities or not. Skipping the orientation discontinuities or not.	
PTS point cloud (*.pts)	Choosing object or UCS coordinates. Setting a subsampling factor for the exported cloud. Choosing the saved number of decimals. Choosing whether to export a color layer and which one. Skipping the depth discontinuities or not. Skipping the orientation discontinuities or not.	

Meshes

Export format	Supported options	Limitations
3DS (*.3ds)	Supports mesh textures.	It is not possible to save meshes with more than 65536 vertices or more than 65536 triangles.
AutoCAD data exchange format (*.dxf)		Does not support mesh color or mesh textures.
COLLADA (*.dae)	Supports mesh textures.	
PLY mesh (*.ply)	Selecting whether to export a color layer and which one. Selecting whether to export local or UCS coordinates. Selecting whether to export point normals or not. Selecting the storage mode among ascii, binary little endian, binary big endian. Selecting the coordinate precision between float and double.	If the model to be exported has too big UCS coordinates (more than 100 km), Reconstructor® will not export it in UCS coordinates, because of PLY format limitations. Does not support mesh color or mesh textures.
STL mesh (*.stl)		Does not support mesh color or mesh textures. Only local coordinates are saved, the model global

		positioning is lost.
Triangle Meshes (*.rtm)		
Triangle Meshes Text File (*.txt)	Selecting whether to export local or global coordinates. Choosing the saved number of decimals.	
VRML (*.wrl, *.vrml)	Supports mesh textures.	
Wavefront OBJ (*.obj)		

Export E57 point cloud format

The E57 point cloud format allow you to add some useful information to the exported object.

First of all, it's possible to export the point cloud as

- E57 grid point cloud (*.e57)
- E57 unstructured point cloud (*.e57)

according to the original data in the project.

Export data layers

There is the possibility to export the the Reflectance color layer and another layer chosen by the user from those available. Also the color for invalid points can be exported, if desired.

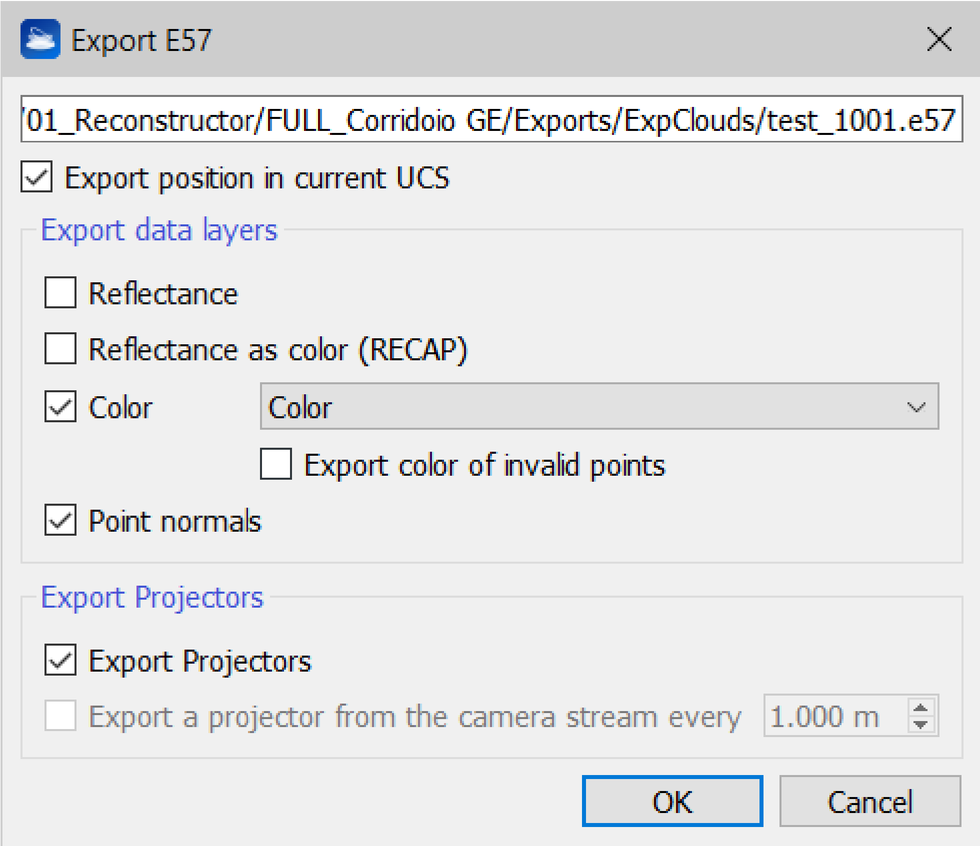
Point normals

A surface normal vector is added to each point to indicate the surface orientation.

Compared to other export format one main feature is added.

Export Projectors

- **Export Projectors**: by flagging this option you can export panoramic projectors that are "children" of the point cloud, as equirectangular 2D image embedded in the .e57 point cloud, keeping the same relative position cloud-camera as in the Reconstructor project.



Export polyline

This option allows you to export a set of polylines to the DXF format, to be read for example in AutoCAD. When selecting this function, you are asked to select the output filename. After that, you are asked to provide the decimal precision that the polyline's points should have.
You can specify a distinct output filename and decimal precision for each polyline. Otherwise, in the select file dialog you can check *Do not ask again* and you will set the decimal precision only once for all the polylines. The filename will be also assigned automatically.

Export cross section as

This option allows you to export a set of cross sections to the DXF format, to be read for example in AutoCAD. When selecting this function, you are asked to select the output filename. After that, you are asked to provide the decimal precision that the cross sections' points should have.
You can specify a distinct output filename and decimal precision for each cross section. Otherwise, in the select file dialog you can check *Do not ask anymore* and you will set the decimal precision only once for all the cross sections. The filename will be also assigned automatically.

Export UCS to AutoCAD

This function exports an User Coordinate System to AutoCAD. The UCS is saved as a .SCR file, a script that can be loaded in AutoCAD.
To import an UCS in AutoCAD, type _script, enter, select the script file and open it. The imported UCS is set as current and the top view is applied.
Or drag'n'drop the *.scr file into the cad drawing to recreate the UCS.

Export annotations

This function exports a set of point annotations obtained with the Annotation tool.

After selecting one or more *Annotations* in the Project Window, use the *Export* > *Export Annotations* command to exports a list of points (in *.txt, *.csv, *.tsv formats) including labels and coordinates (in the current UCS).

In the exported file, your annotation will be organized as the example below.

Annotation ID	*Annotation Name*	*X*	*Y*	*Z*	*Color mapped layer*	*Value (*)*	*Comment*	*Hyperlink*
880	Reflectance: 0.380	6.257	3.568	2.697	Reflectance	0.380	880	Grids/test_1000_1_sub2.rgp
881	Reflectance: 0.428	6.473	1.084	2.232	Reflectance	0.428	881	Grids/test_1001_1_sub2.rgp
882	Reflectance: 0.382	6.496	6.411	2.690	Reflectance	0.382	882	Grids/test_1000_1_sub2.rgp
883	Annotation (1)	5.797	4.309	2.694	Inclination	0.023 0.026 -0.999	883	Grids/test_1000_1_sub2.rgp
884	Annotation (2)	7.745	3.630	2.698	Inclination	-0.026 -0.002 -1	884	Grids/test_1000_1_sub2.rgp
885	Annotation (3)	7.682	6.204	2.694	Inclination	0.001 -0.037 -0.999	885	Grids/test_1000_1_sub2.rgp

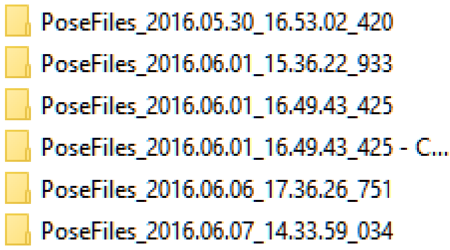
(*) one value for scalar color layers (e.g. reflectance), three RGB components for not scalar layers (e.g. color, inclination).

Export pose

This is a very useful function for registration workflows. It allows to safely store in a separate folder positions and orientations of your project items, in order to recover them if and when needed.

When pre-registering, registering, geo-referencing or moving around your models, you may want to backup the model's positions and orientations (poses) before changing them. If something doesn't work in the registration algorithm, or in changing the models' poses, you can safely restore the poses as they were before the action.

When you activate this function, the poses of the selected items are saved in the Results/poses folder, in a folder called with the name pattern *PoseFiles_aaaa.mm.dd_hh.mm.ss_xxx*. This way, you can easily read the precise date and time of creation of the poses, for future use.



To restore any item's pose, use the pose transform dialog.

Export to Autodesk® ReCap®

Reconstructor® can directly export point clouds in a .rcp project format for a more complete and faster integration with the Autodesk® ReCap® software.

After selecting the point clouds in the Project Window, you can directly select the *Export > Export to Autodesk® ReCap®* command from the contextual menu (or from *File* > *Export* menu) .
The following dialog displays the list of selected point clouds.
For each of them you can select the point cloud's *Color layer to export* which will be exported as "RGB" of the "Color Mode" of ReCap®. "Reflectance" and "Inclination" color layers, if present, will be automatically exported in the respective "Intensity" and "Normal" color modes of ReCap®.
After confirmation, you have to select the destination directory where the project files "recap_project.rcp" and a "recap_project Support" subfolder (including .rcs files of the individual clouds) will be saved.
When export process is finished, a successful message will appear, giving you the possibility to directly open the export folder.

There is no limit on the number of exported clouds.

When exporting structured clouds with/without associated camera, the RealView visualization is available; take note that you cannot see the invalid points colorization.

No ReCap® license is required on the same PC.

The command is not available for Trial and Viewer versions of Reconstructor®, as all export commands are prevented.

Navigation menu

Reconstructor® provides many interaction styles to navigate in the 3D scene.
The navigation modes are selectable through the *Navigation toolbar* (placed on the right, most used commands) or by *Navigation* command in the top menu and are grouped by type:

- Predefined views
- Navigation modes
- Screen settings
- Create UCS from point list

In the contextual menu of items other commands are available:

Go to
To center the 3D window to the bounding box of the model. The view point is computed so that to contain the whole bounding box in the viewport. Warning: this could place sometimes the model far away, with the effect that it seems not visible. Therefore increase the max depth in the Options.
This tool is very useful when you have loaded many objects and want to go quickly and easily to a specific one.

Center to local origin
The view point is aligned to local coordinate frame of the model, with X rightwards, Y upwards and Z going out of the screen.

Align to bounding box
The view point is aligned to the desired face of the bounding box of the model.

In View Parameters dialog you can customize advanced parameters of the current view by controlling how the "virtual eye" sees the 3D scene with all your models.

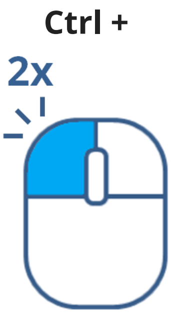
After creating an item, it is highlighted and focused in the Project window and in the 3D window. If the item is included in a group, the group will be automatically opened. (e.g. after point creation with Alt+double-click LMB the point is inserted and automatically focused in Project window).

Shortcuts

To focus and zoom in on a particular item you can:

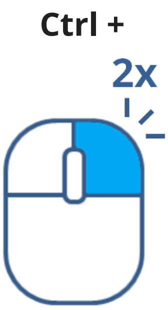
- select the item in the Project Window and use the *GoTo* command in the contextual menu
- select the item in the Project Window and double click with the scroll wheel anywhere in the 3D window
- select the item in the Project Window and use the *Predefined Views>Default View* command

Double-clicking with LMB (primary button) on a point in the 3D window, the item to which the selected point belongs is highlighted in the 3D window and in the Project window. If included in a group, the parent groups are automatically opened.



Holding **Ctrl + double-clicking with LMB** (primary button) on a point in the 3D window, the item to which the selected point belongs is added in a multiple selection, highlighted in the 3D window and in the Project window. The 3D view is focused on the last selected item. If included in groups, the groups are automatically opened.

Double-clicking with RMB (secondary button) on a point in the 3D window, the item to which the selected point belongs is highlighted in the 3D window and in the Project window, while the context menu appears.



Holding **Ctrl + double-clicking with RMB** (secondary button) on a point in the 3D window, the item to which the selected point belongs is added in a multiple selection, highlighted in the 3D window and in the Project window. The 3D view is focused on the last selected item , while a common context menu appears.

Predefined views

Default view

This navigation mode allows the user change the current view so that all the selected items are completely visible in the 3D scene.

Bubble view

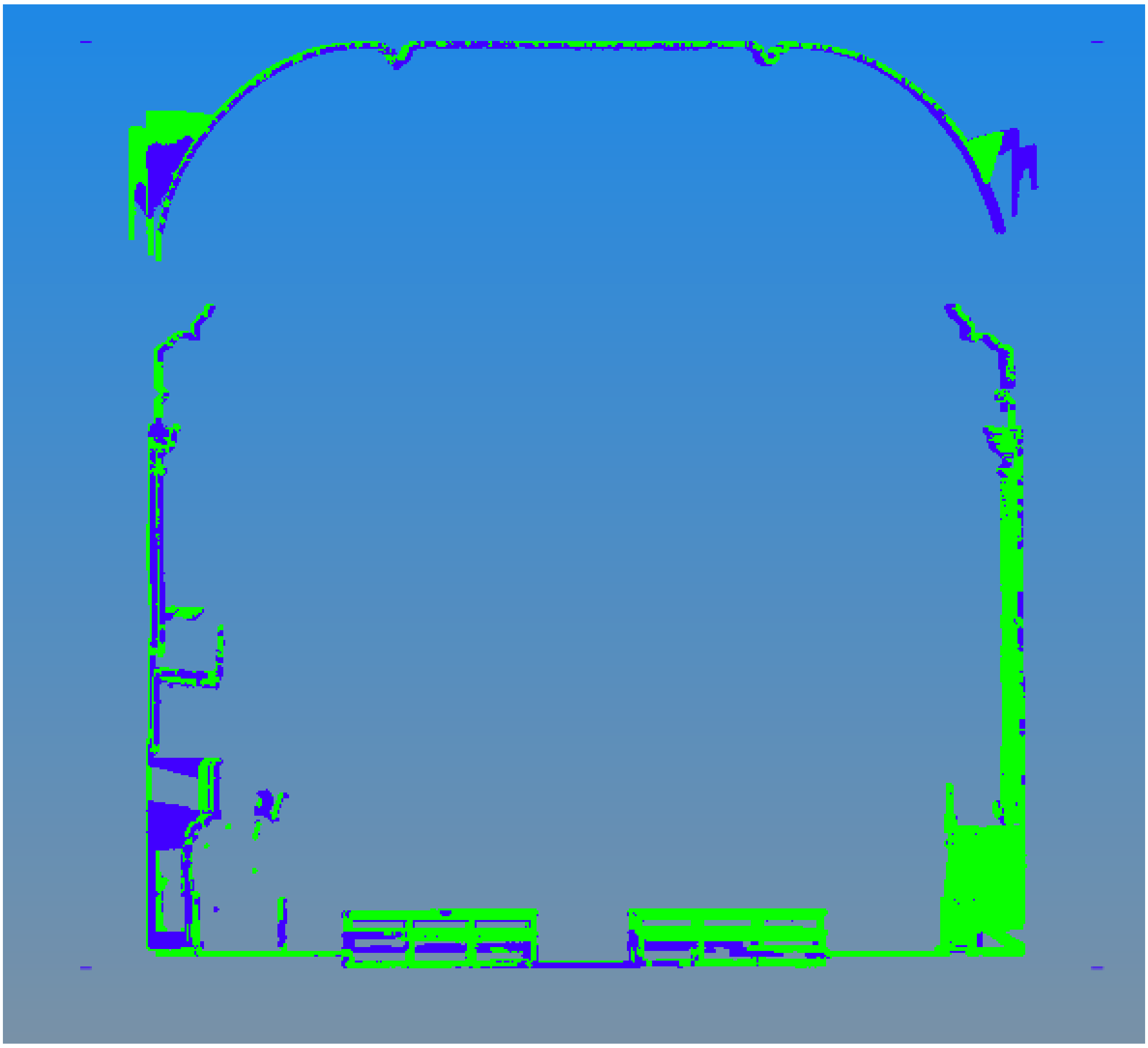
This navigation mode allows the user to place the current view to the center of an item. Only camera rotations are allowed in this mode.
See also Bubble mode.

2D view
This navigation mode opens the Edit 2D Grid Window. Note that this view is supported for grid point clouds only.

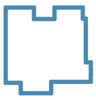
Map view
This navigation mode places the view so that all the selected items can be seen from a top view.

Slice view
This navigation mode automatically switches the view to orthographic mode and alignes it to the selected items bounding box vertical plane.

This mode can be used to check the correct alignment along a vertical section of the displayed items. The depth of the slice can be changed from the Display and navigation Options dialog.



Keyboard/mouse movement	Effect
LMB + move mouse	Rotate the around the selected items vertical (z) axis
CMB + move mouse	Translate sideways
Mouse wheel rotation	Translation towards point indicated by the mouse

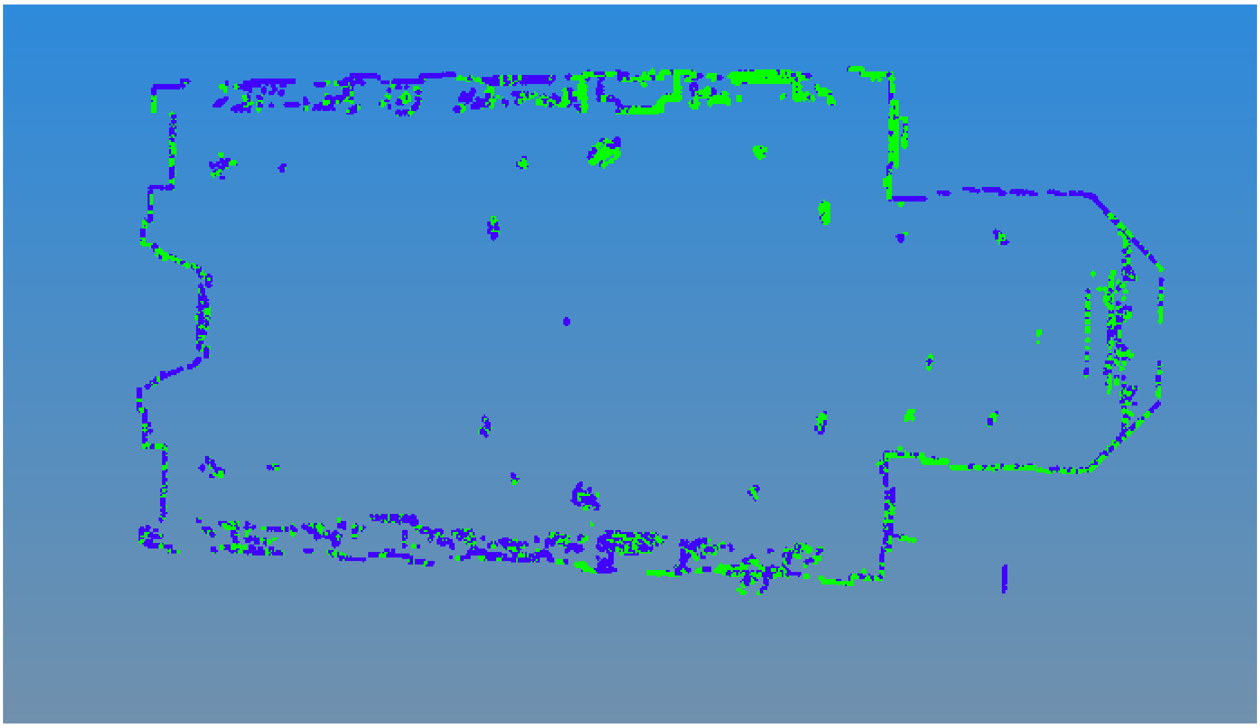


Horizontal Section View

This navigation mode automatically switches the view to orthographic mode and alignes it to the selected items bounding box horizontal plane.

This mode can be used to check the correct alignment along an horizontal section of the displayed items.

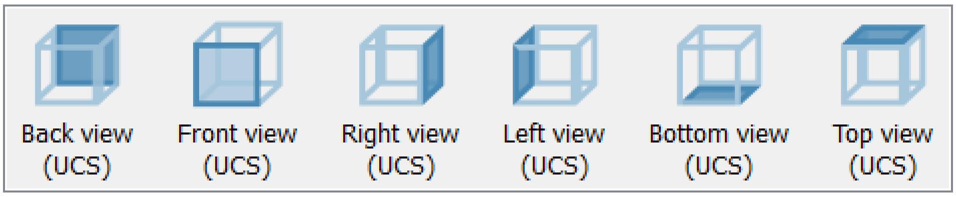
The depth of the horizontal slice can be changed from the Display and navigation Options dialog.



Keyboard/mouse movement	Effect
LMB + move mouse	Rotation the around the selected items vertical (z) axis
CMB + move mouse	Translation sideways
Mouse wheel rotation	Translation towards point indicated by the mouse

UCS views

This navigation mode places the view so that all the selected items can be seen from an according to UCS view:



Navigation modes

Pivot mode
Default navigation mode based on translating and rotating around a chosen point of the 3D scene

Walk mode
Navigation mode that allows you to translate and rotate around yourself, independently of the models

Bubble mode
Navigation mode in which only rotations around yourself are allowed

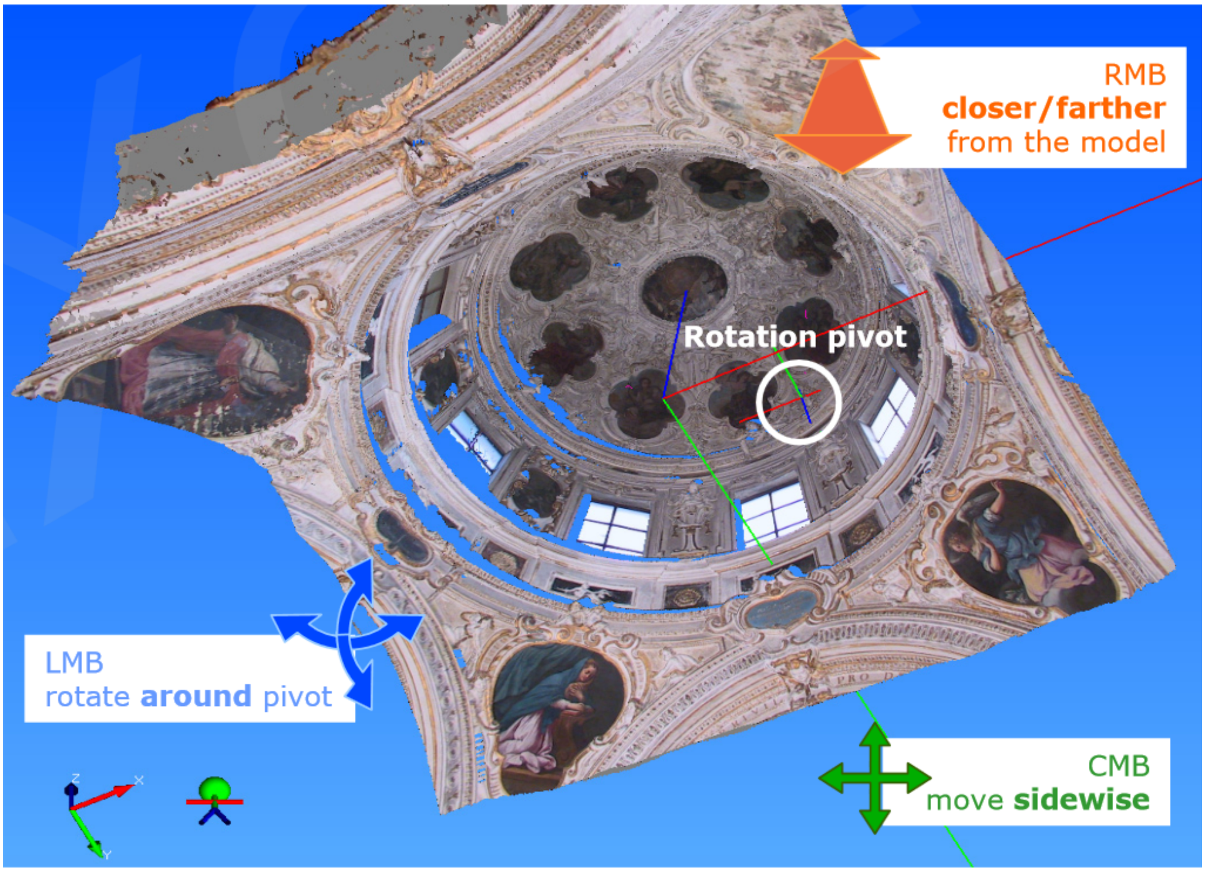
Zoom
Narrows down the virtual camera's field of view to concentrate on a particular viewport. It doesn't move the virtual camera

Move to viewport
Moves the virtual camera so that a particular viewport is in sight, without changing the field of view

Recent camera positions
Recovers up to 30 steps you went through during your navigation

Pivot Mode

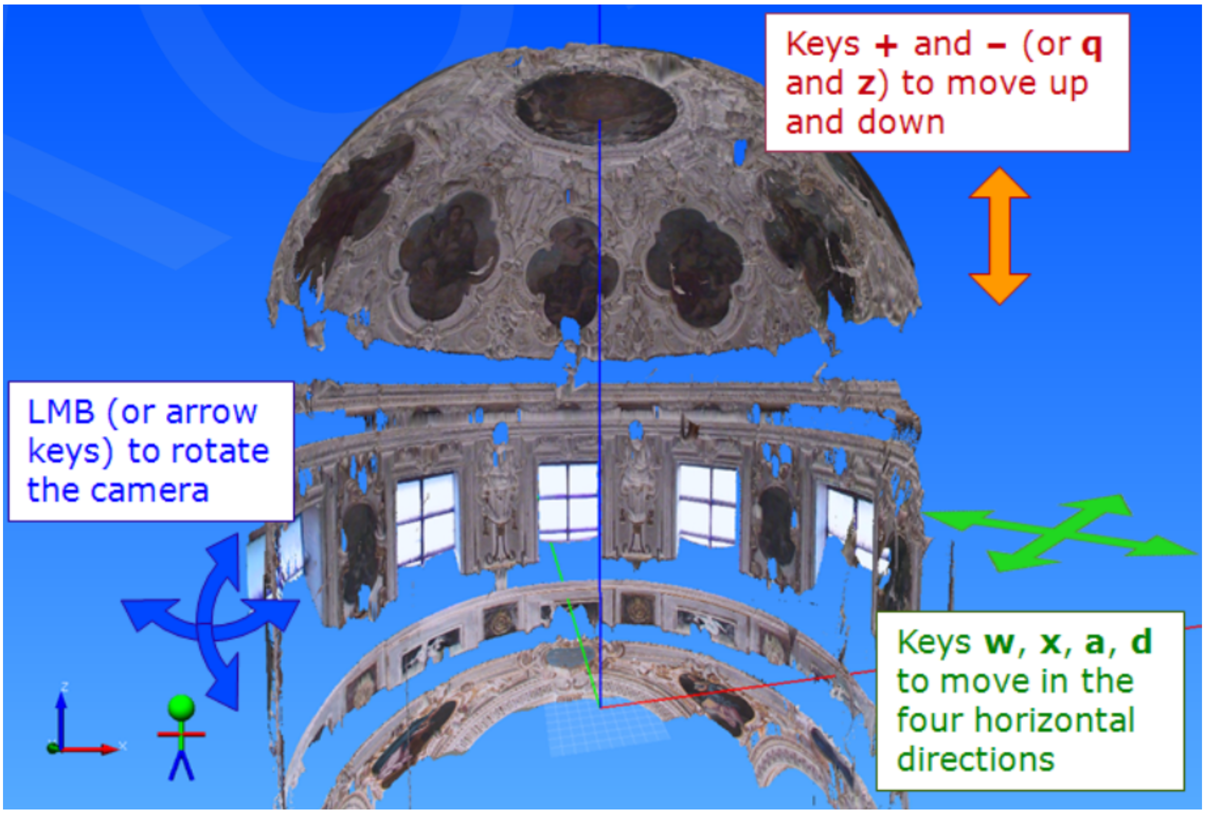
In this mode a point can be selected as a rotation pivot. *The pivot is set by a left button click on a valid point of an object*.



Keyboard/mouse movement	Effect
LMB + move mouse	Camera rotation around the pivot
CMB + move mouse	Camera sideways translation, pan
RMB + move mouse back/forth	Camera dollying, forward and backward along the current focal axis
Mouse wheel rotation	Translation towards point indicated by the mouse pointer
Shift + **LMB** + move mouse left/right	Camera rolling
LMB click on a model's point	Selection of rotation pivot
Shift + mouse wheel rotation	Zoom in and out (change of field of view)
Alt + **LMB** double click	Selection of points for point list window
Shift + **CMB** + mouse move	Draw rectangle to move into

Walk mode

This mode is designed to simulate a human being walking around the 3D scene. The user can make translation movements with the keyboard and mouse movements with the mouse.



Keyboard/mouse movement	Effect
LMB + move mouse	Camera rotation
w, x, a, d keys	Move forward, backward, left, right
q, z or **+, -** keys	Move up, down

Bubble mode

This mode is designed to simulate a human being walking around the 3D scene. The user can make translation movements with the keyboard and mouse movements with the mouse.

Keyboard/mouse movement	Effect
LMB + move mouse	Camera rotation (pan and tilt)
Mouse wheel rotation	Zoom in and out (change of field of view)
Shift + **LMB** + move mouse left/right	Camera rolling
Alt + **LMB** double click	Selection of points for point list window
Key 'c'	Copy readout window's contents to clipboard

Zoom Tool

This tool allows the user to zoom on a specific part of the scene.

The tool can be used by following these steps:

- Press the "zoom" command under the Navigation menu.
- Drag the mouse while pressing the left mouse button to draw a rectangle in the 3D view.
- Release the mouse: the camera zooms into the new defined viewport. The system then changes back to the last navigation mode used.

Difference with respect to the Move to viewport *tool*
With the zoom tool the camera is zoomed-in a detail of the scene. Only the field of view (in perspective mode), or the projection size (in orthographic mode) is changed.
With move to viewport, the camera is translated closer to the detail indicated by the rectangle, leaving the field of view unchanged.

Move to viewport

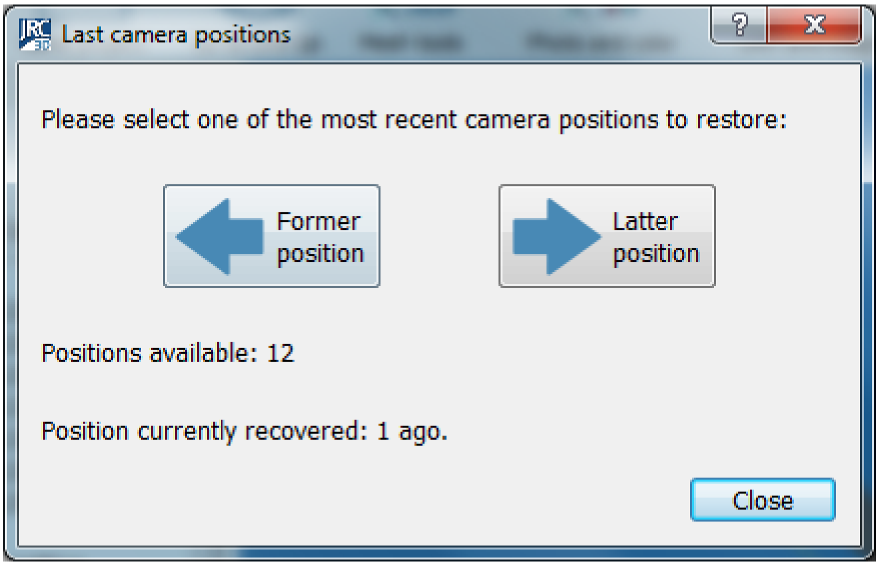
This tool moves the viewport to a specific part of the scene.

The tool can be used by following these steps:

- Press the "move to viewport" command under the Navigation menu.
- Drag the mouse while pressing the left mouse button to draw a rectangle in the 3D view.
- Release the mouse: the camera translates the new defined viewport. The system then changes back to the last navigation mode used.

Difference with respect to the Zoom *tool*
With the zoom tool the camera is zoomed-in a detail of the scene. Only the field of view (in perspective mode), or the projection size (in orthographic mode) is changed.
With move to viewport, the camera is translated closer to the detail indicated by the rectangle, leaving the field of view unchanged.

Recent camera positions

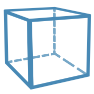


When navigating your models in the 3D scene, it can happen that your make mistakes, you lose your models, or you lose the position you reached. You can use this function to “go back to your steps”.

Reconstructor® keeps in memory the last 1000 camera positions through which you navigated. Through this dialog, you can browse them and go back to your previous steps, restoring useful views on your models when you lost them.

Screen settings

Show axis
To show/hide the coordinate system frame of the current camera global reference (CCGR) in the bottom left corner of the 3D scene.



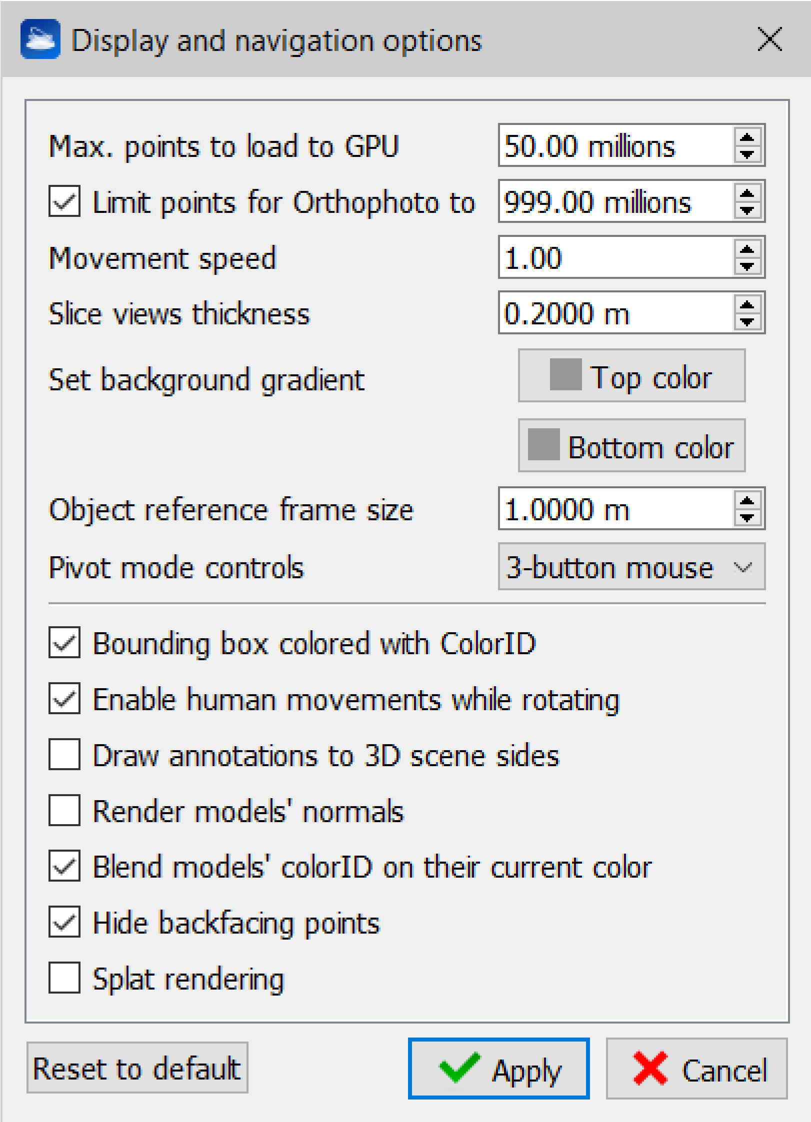
Show bounding box
To show/hide the bounding boxes of all items in the 3D scene.

Color by ID
To render all the project entities with their identifying color ID. The color ID of a point cloud can be changed from the property window.
This option is useful to distinguish the models from each other and to know which model is which in the 3D scene.

(Display and navigation) Options
To modify current navigation and rendering parameters.

Light settings
To set shadow and lightening options.

Display and navigation options



Max points to load to GPU (millions)

This option is used to set a maximum number of point to be rendered in the 3D scene. The limit is cumulative for all the loaded point clouds. It depends on the characteristics of the graphics card.

Each loaded cloud is divided into 200,000 points buffers and, based on the user points budget, the number of buffers to be rendered is calculated.
It is always ensured that at least one buffer for each cloud is rendered. So it never happens that the user does not see clouds loaded, but, on the other hand, it is possible that the maximum budget set by the user (Max points to load to GPU) is exceeded.

Limit points for Orthophoto to ... (millions)

Check this option and set a value to limit the maximum number of points (belonging to the flagged and loaded point clouds) used snapping an orthophoto (or Virtual Scan). It depends on the characteristics of the graphics card. This limit is not concerning the X-ray orthophotos and Blueprints.

Movement speed

This option sets the translation speed when panning inpivot mode. It is an adaptive parameter, for the most part the default value (1) should be fine.

Slice views thickness

This option sets depth of a single slice when using the Slice view or the Horizontal section view.

Set background gradient

Change the uppermost and lowermost color of the background of the 3D rendering window.

Object reference frame size

All models and cameras can draw their local coordinate frame by enabling *Draw coordinate frame* property in the Property Window. Here the frame size (in meters) is specified to improve its visibility accordingly to the extent of the scene.

Pivot mode controls

Here you can select two options: "*Reconstructor*" and "*3-button mouse*". These options are two sets of controls for the Pivot Mode. The second one is the default and is described here.
If you select "*Reconstructor*", then in pivot mode the following controls apply:

- Rotate: To rotate the model around the pre-defined center of rotation (pivot), move the mouse within the 3D window while keeping the left button pressed.
- Pan: To pan the model in X and Y, move within the 3D window while keeping the left button and the Shift key pressed.
- Zoom: To translate the model along the Z-axis, move the mouse within the 3D window while keeping the left button and the [Ctrl] key pressed or use [Ctrl + mouse wheel] to zoom in and out along the Z-axis. In the latter case, if the mouse pointer hovers a valid 3D point, the zoom converges to that point.
- Change pivot: double-click the left mouse button plus [Ctrl+Shift] on a valid 3D point.

Bounding box colored with colorID

When this option is checked, the bounding boxes of the project items are colored based on the item's colorID. If this option is unchecked, the bounding boxes are colored with the default (yellow) color.

Enable human movements while rotating

When this options is checked, a small human avatar appears in the bottom-left corner of the 3D rendering window. While you navigate in the 3D scene, you will experience that after doing many rotations, your viewing direction will still be oriented according to the vertical axis and to the horizontal plane defined by the current UCS. This is like a human being that, after rotating his head around his neck in many directions, when relaxing the head will go back into the natural position, which is in line with the spinal chord.
When this option is unchecked, while navigating the 3D scene you will experience that after some rotations the reference to the UCS's horizontal plane is lost, and that you are floating in the space like an astronaut floats among meteors, without gravity force.

Draw Annotations to 3D scene sides

Annotations will be arranged on the sides of the render window, at the exact insertion point.

Render models' normals

View vectors that describe the normals directions for point clouds and meshes, if they have been computed.

Blend models' Color ID on their current color

It blends the color layer (reflectance, confidence...) whit the Color ID. This two features are available in the Property Window.

Hide backfacing points

This setting hides the points that are oriented in the opposite direction to the current point of view. The orientation is calculated according to the point normal. The surfaces scanned from a point opposite to the current point of view will be hidden.
For example, if you scan a room, the ceiling seen from above (therefore from outside) is made transparent and you can see the inside of the room.

The effect is equivalent to the *Cull face* of meshes, but applied to point clouds.

This setting has no effect if the clouds do not have the normals.

Splat rendering

This setting enables / disables the rendering of point clouds through circular elements (splats) with the aim of reconstructing the surface of the point clouds, covering the "holes" given by discrete sampling.

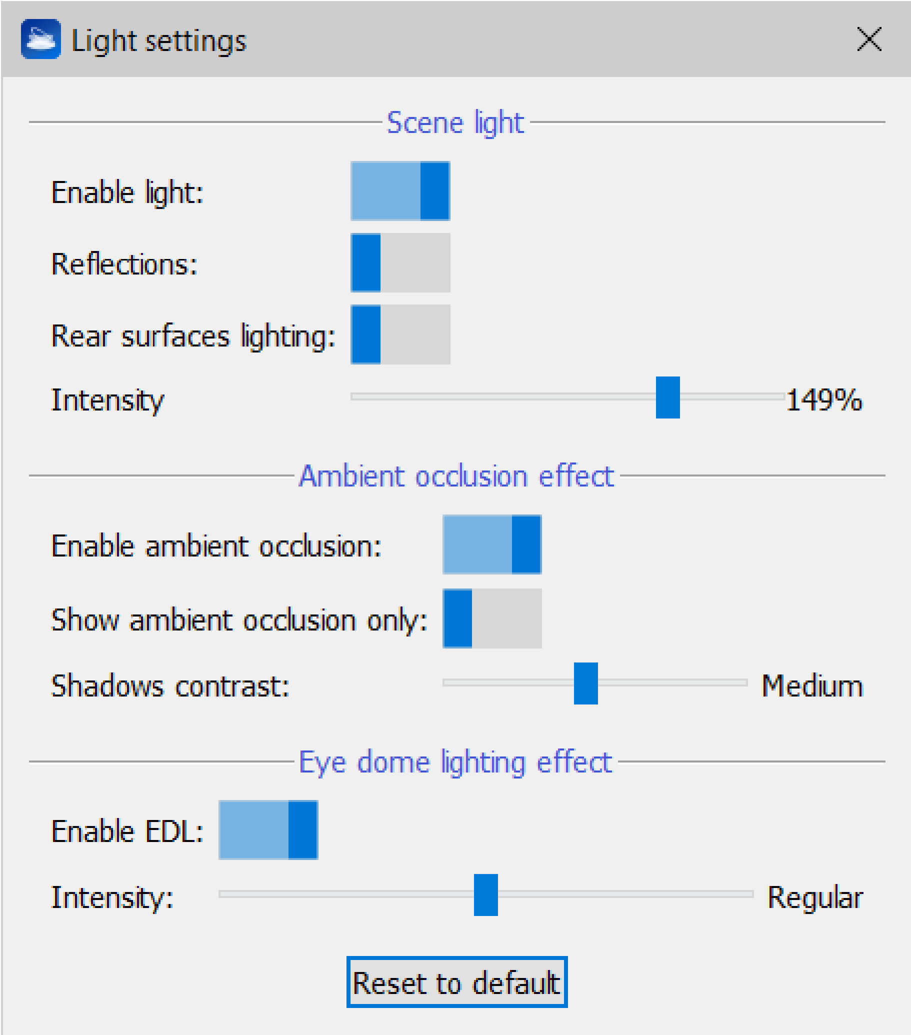
The point cloud *normals* must be calculated for splat rendering to take effect. It is also possible to make the rendering more precise by calculating the radius of influence of each point using the Point RadiusExtraction filter. The radius of the splat also depends on the percentage of points rendered.

Click on *Reset to default* button to reset to default parameters.

See also Light Settings options.

Light settings

The 3D visualization of point clouds and meshes can be modified changing the Light effects settings.



Scene Light

Enable light

This setting enables / disables all light effects in 3D scene. Disabling light effects also lets the application to use less graphics and memory resources.

Reflections

This setting enables / disables the specular components of the projected light. This setting can be useful to enhance some fine details while navigating the rendered models.

Rear surfaces lighting

This setting enables / disables the lighting of surfaces that have normals opposite to the point of view direction.
The *Intensity* changes the overall strength of the 3D scene light.

Ambient occlusion effect

Enable ambient occlusion

This setting enables / disables a special rendering effect which attempts to simulate the shading effect of a light coming from the user perspective by computing light attenuation due to occluded volumes. The resulting model shading is a blend between the model current color layer and the computed light attenuation factor.

Show ambient occlusion only

This setting renders only the computed ambient occlusion factor (light attenuation). The resulting model color is not blended with its current color layer. This setting is only available if ambient occlusion is enabled.

Shadow contrast

This setting changes the ambient occlusion computing parameters to make the resulting shadow more (or less) intense. This setting is only available if ambient occlusion is enabled.

Eye Dome Lighting (EDL) effect

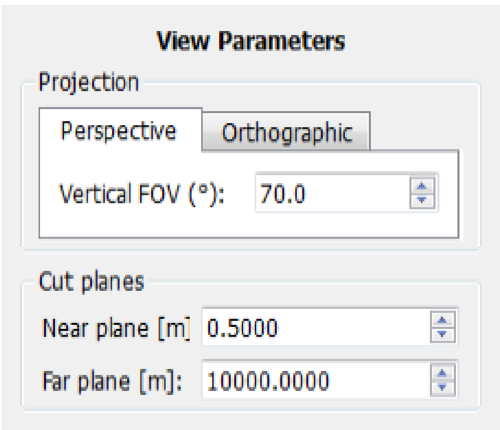
Enable EDL

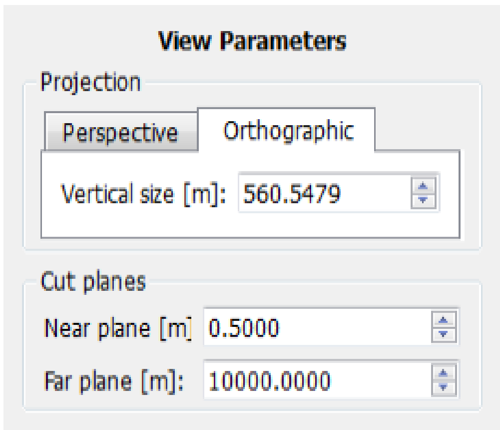
This setting enables the EDL, a lighting technique that highlights the discontinuities of the surfaces making the edges of the displayed surface more or less dark depending on how far the foreground surface is from the background one.
As you increase the Intensity, the edges will be more marked.

Click on *Reset to default* button to reset to default parameters.

View Parameters

This window appears when the user hovers the mouse on the *View parameters* button in the top toolbar. If you click on the button the little window remain visible.
Here the user can customize advanced parameters of the current view.





This window allows you to control how the “virtual eye” sees the 3D scene with all your models. There are two viewing mode: the perspective mode (shown on the left image) and the orthographic mode (on the right image).

Perspective projection

In perspective mode, the virtual eye uses a perspective projection to see the scene, and you can set the amplitude of the projection by specifying the “Vertical FOV” (field of view) in degrees. Decreasing (increasing) the field of view is equivalent to zooming in (out) with the objective lens of a camera.

Orthographic projection

In orthographic mode, you see the 3D scene through an orthographic projection, and you can specify the “vertical size” in meters of your projection.

Cut planes

Near plane: distance of near clipping plane in meters. Specifies the minimum rendered depth. In perspective mode only values greater than zero are allowed.
Far plane: distance of far clipping plane in meters. Specifies the maximum rendered depth. Must be greater than Near plane.

To optimize the depth accuracy try to keep these values as close as possible to the desired scene depth range. If your model's bounding box is 100 x 100 x 100 m, it is not useful to adopt a far plane of hundreds of kilometers.

Auto Adjust

If flagged, near and far plane are automatically computed, based on the distance between camera and pivot: the near plane is set at 1/100 and the far plane is set at 50 times the camera-pivot distance. It only applies to perspective mode.

You can save the current settings by the Place here a survey point function.

Data management

In Reconstructor® several processing tools are achievable and grouped by:

- **LineUp**
- **Selection & Editing**
- **Cloud processing**
- **Meshing**
- **Geometric shapes**
- **Photo & Color**

You can find most used tools also in the Top Toolbar.

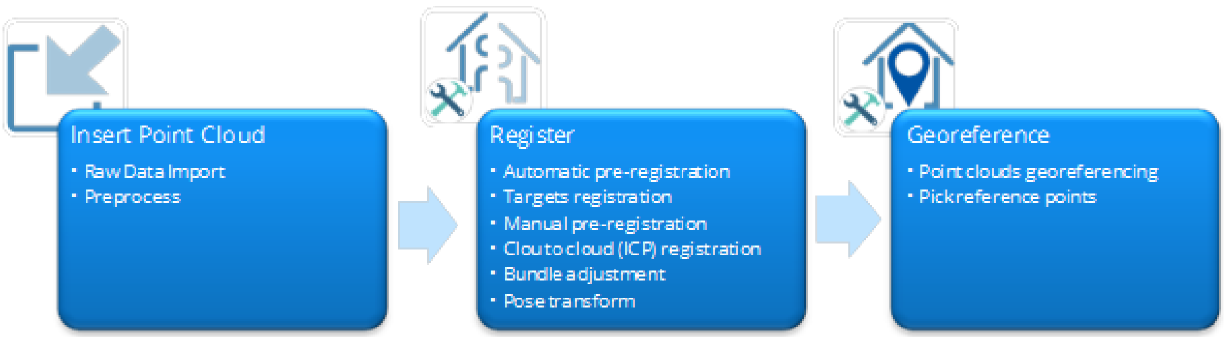
LineUp®

LineUp® is a tool in Reconstructor® that easily allow you to:

- Import
- Preprocess
- Register
- Georeference

any set of points clouds (both unstructured and structured -grid- point clouds) and meshes.

In general, your workflow is assisted by following the steps here illustrated:



In the Top toolbar you can find the most used commands, grouped by main processes:

Save Project
This function saves the current project, safley storing all the project items, their load/unload state, their properties, the current 3D camera pose, etc.
It is advised to save the project frequently, especially before long processing operations.

Import point clouds
To import every point clouds format supported by Reconstructor®.

Wizard
To open the Scan Processing Wizard and import, preprocess, preregister and register any set of scans you are importing in LineUp® tool.

Cloud-to-cloud (and mesh) registration
To register the imported scans. This command includes both pre-registration process and fine registration

Target-based registration
To register the imported scans by using targets and control points, flat circular targets or spheres.

Georeferencing
To georeference the models by using known control points.

UCS & positioning
To create and edit User Coordinate Systems and the position of the 3D models in it.

Cloud processing
To filter and manage point clouds by merging, simplifying and modifying them.

Targets, annotations and points conversion

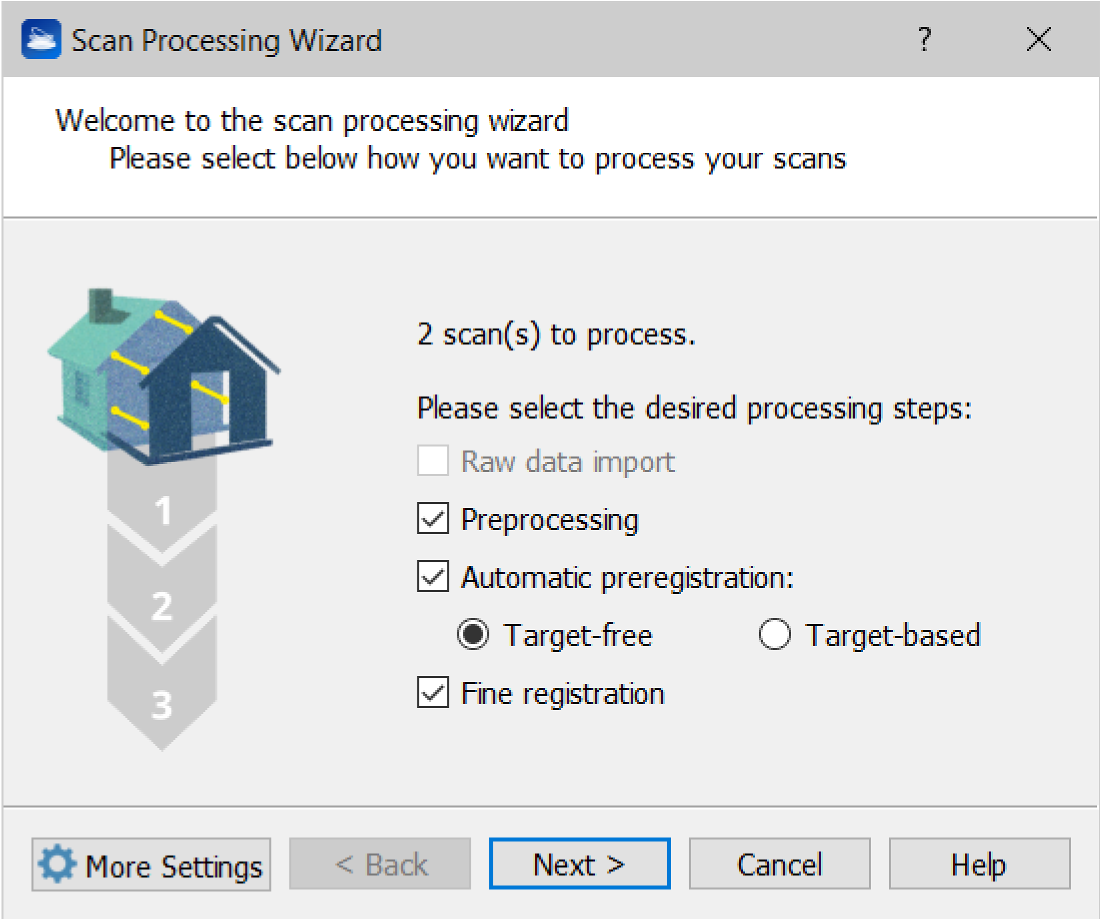
Scans Processing Wizard

You can access it at the beginning of your work, by dragging&dropping the point clouds (you want to import and process) in the 3D window or in a second time, by clicking the *LineUp>Wizard* command.

This wizard allows you to:

- Import
- Preprocess
- Preregister
- Register

any set of scans in LineUp® tool.



You must simply follow the subsequential instruction to import, preprocess and register point clouds.
By clicking on *More Settings* the *Wizard Settings* will appear.

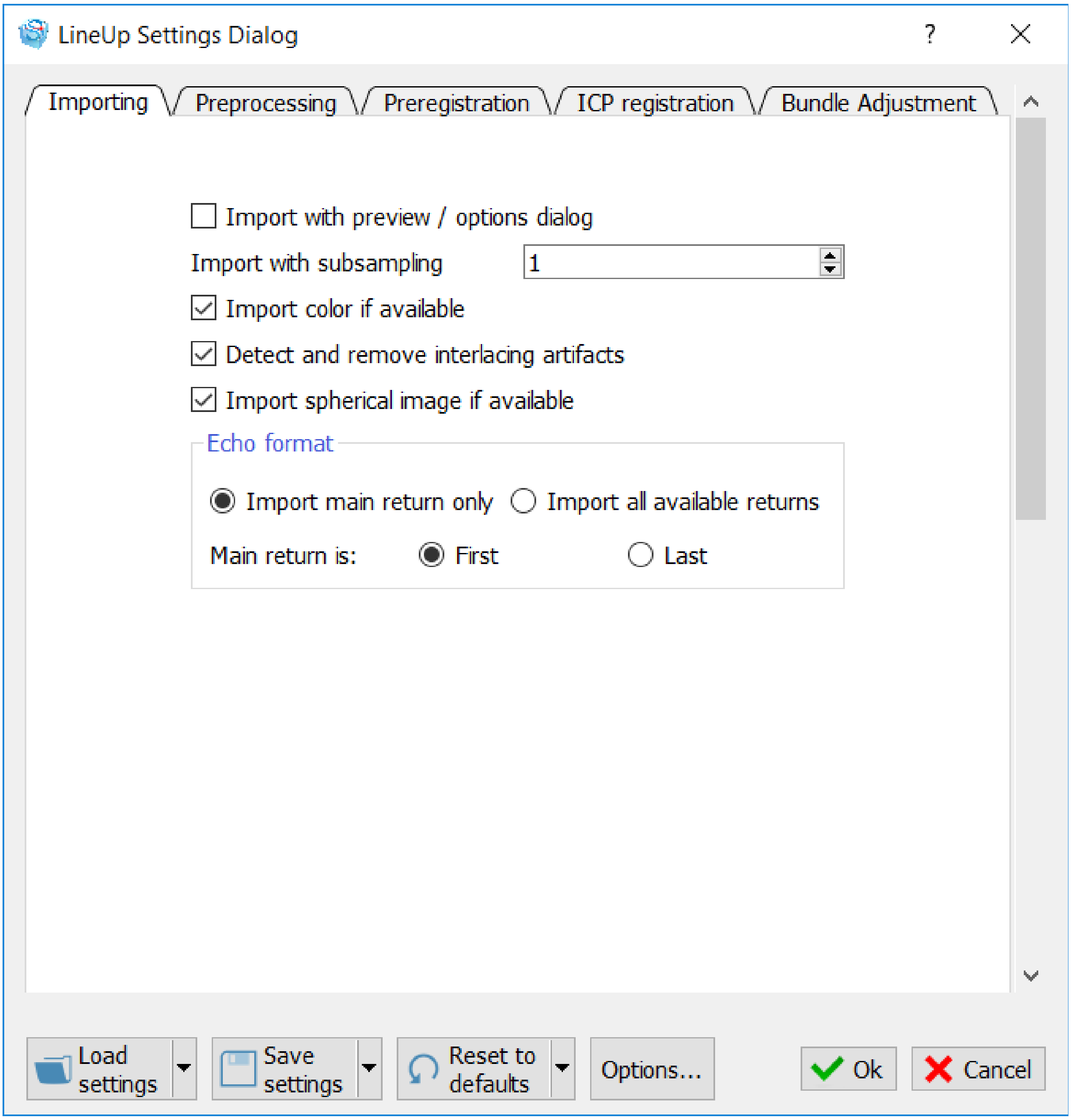
Scans Processing Wizard

By opening it you can directly change the settings used to import, process and register the clouds.
These settings are subdivided in tabs, related to the following processes:

- Importing
- Preprocessing
- Preregistration
- ICP registration
- Bundle Adjustment

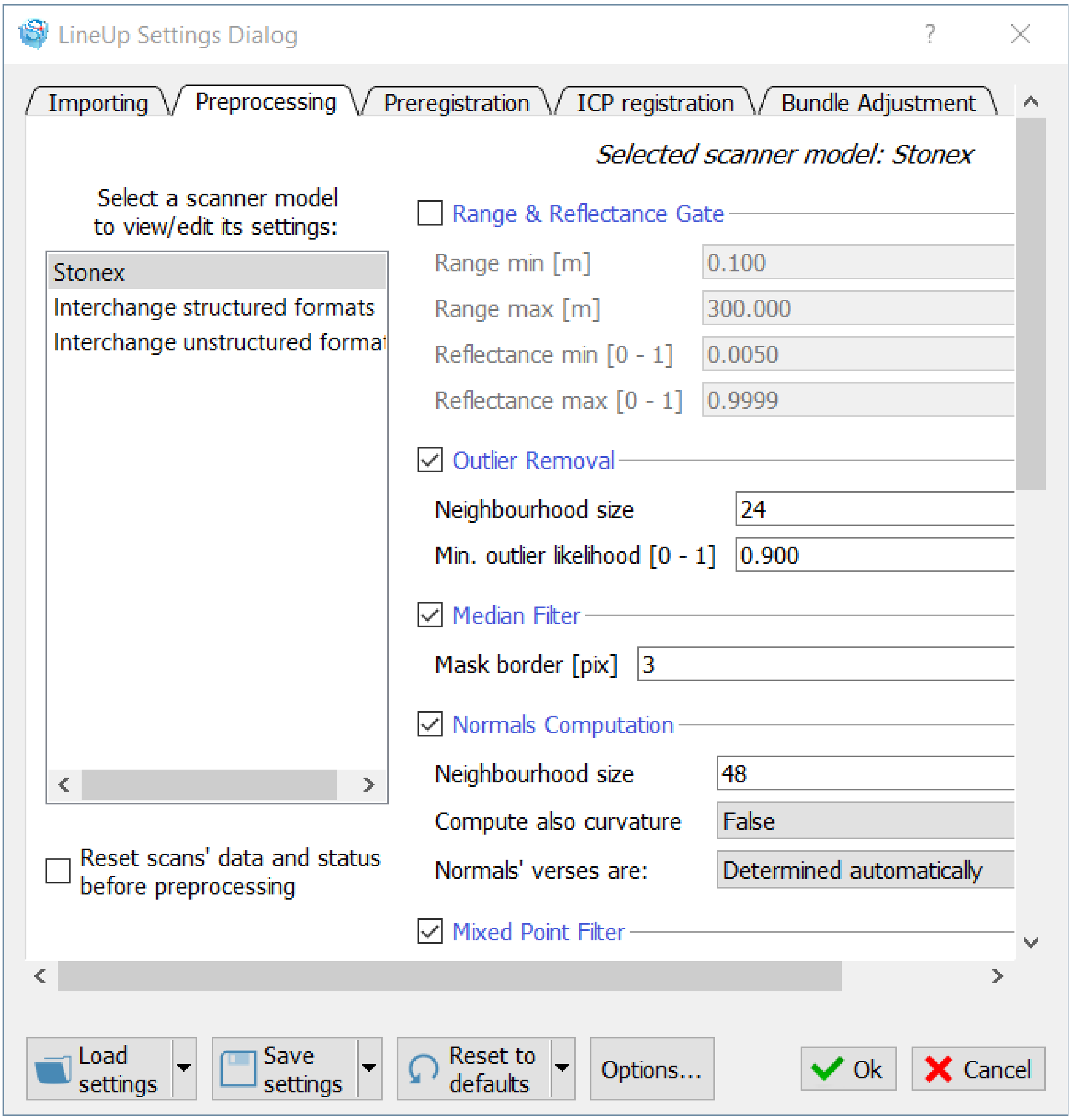
These processes are managed by properly setting the parameters for each analysis step.

Importing

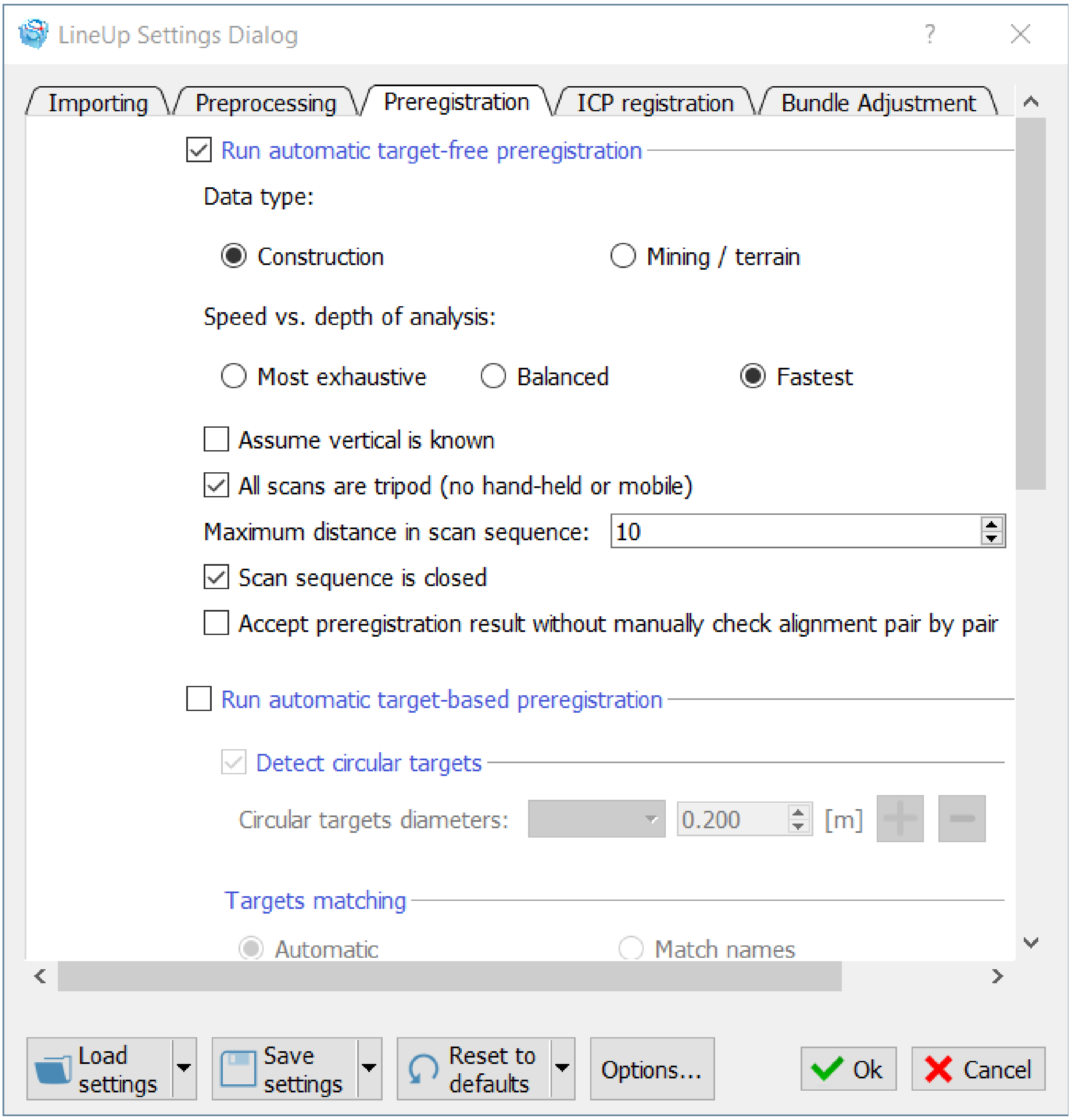


Preprocessing

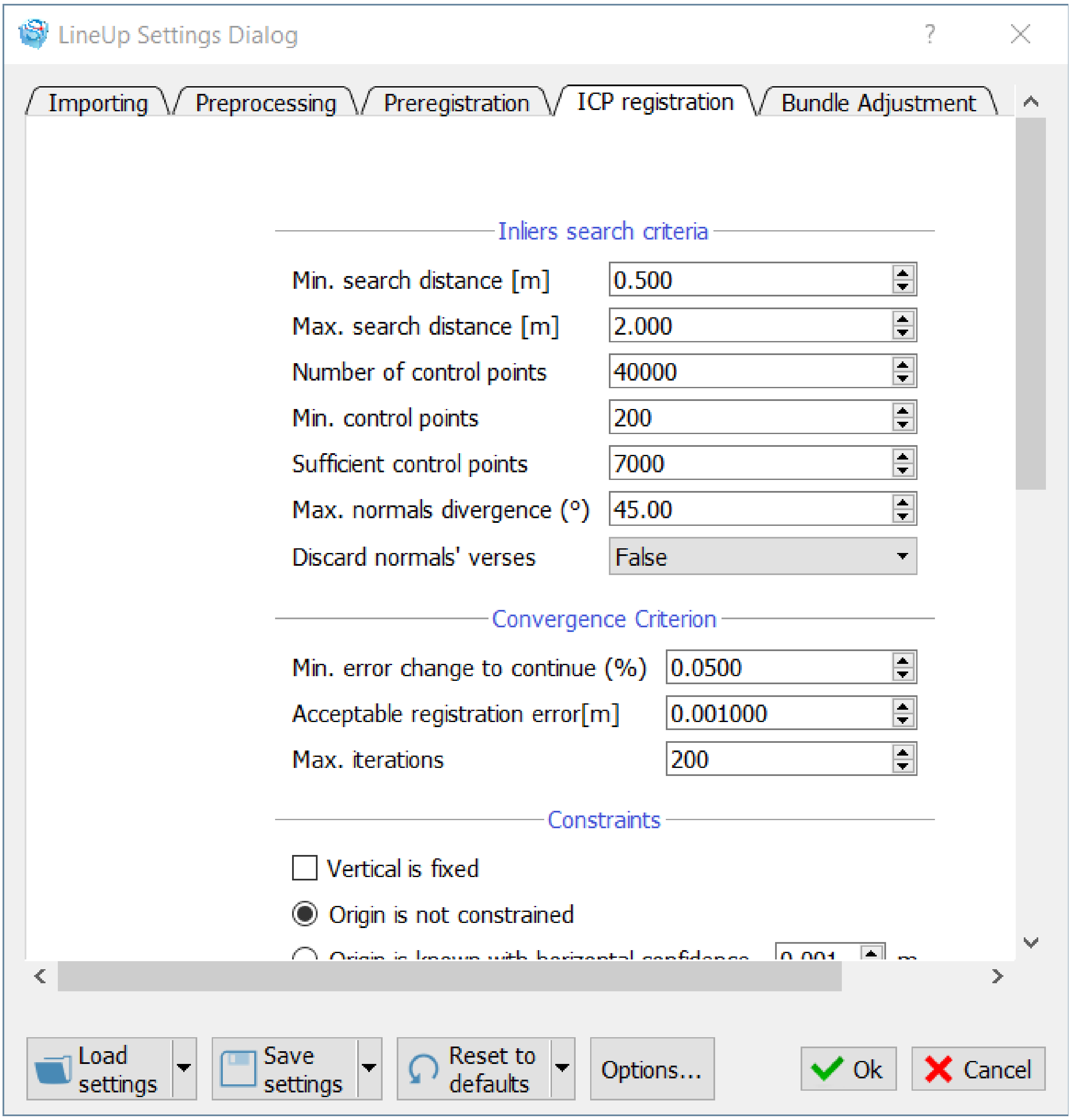
Reconstructor® recognizes the file format and applies default parameters for preprocessing.
If you performed your scans using a Stonex laser scanner, you can leave "Stonex" as scanner model.
The filters are detailed in Pre-processing page.



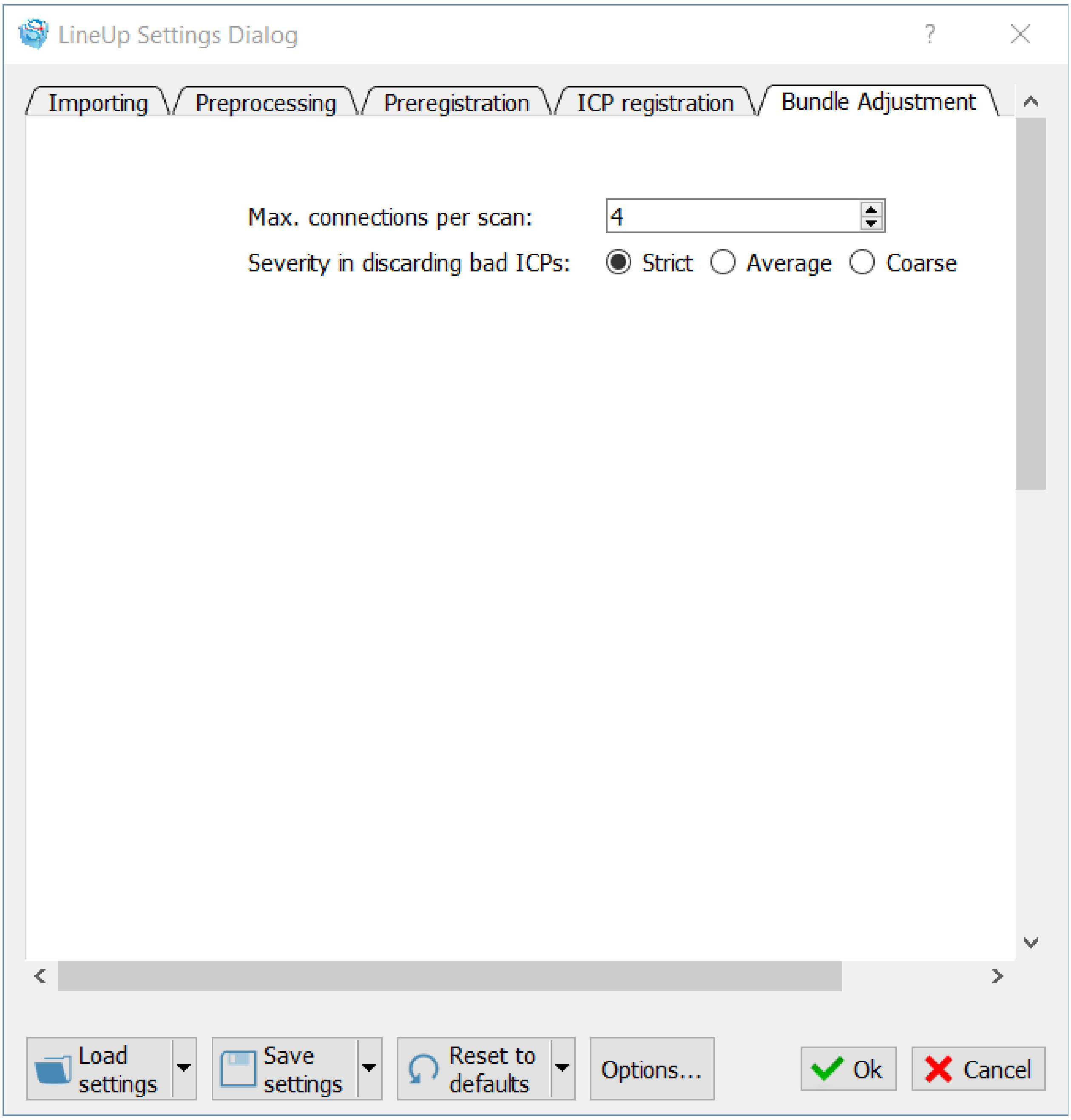
Preregistration



ICP registration



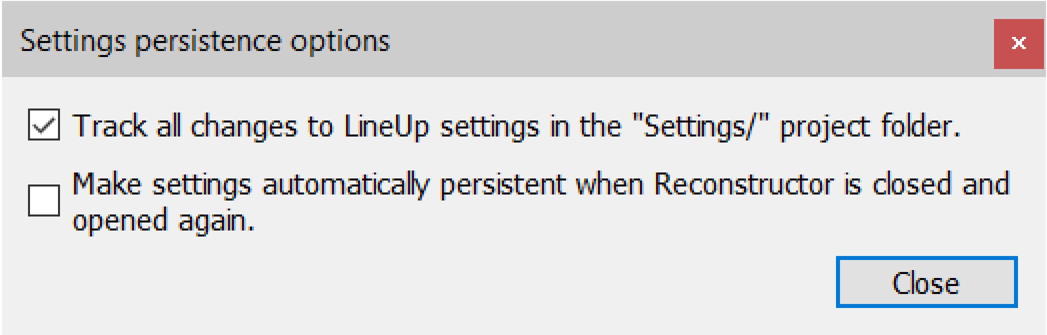
Bundle Adjustment



Manage settings

Through the buttons in the lower part of the Wizard window the parameters settings can be managed.

- *Save settings/Save all tab's settings*: to save current settings in a .xml format (by default in *Settings* folder);
- *Load settings/Load all tab's settings*: to load previously saved (even by a third party) settings in a .xml format (by default the upload folder is the Settings one);
- *Reset to default/Reset all to default*: to reset the current settings to the default ones;
- *Options*

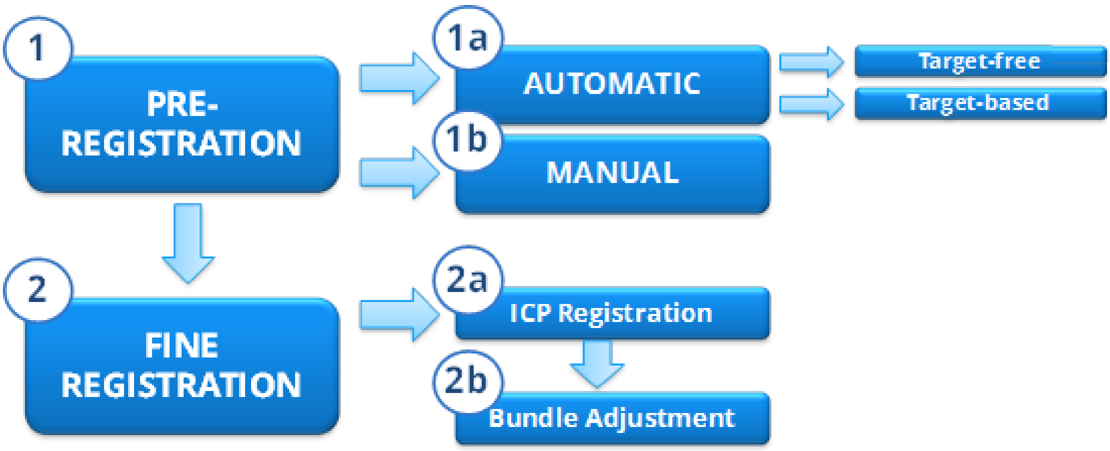


1. Not flagging options: the settings of the current Reconstructor® session are kept and they are temporarily saved in registry keys. Closing and reopening Reconstructor® the parameters return to default values.
2. Flagging only the first option: the settings of the current Reconstructor® session are kept and the parameters are saved in the *Settings* folder (.xml) at each modification. Closing and reopening Reconstructor® the parameters return to default values, but keeping the possibility to load previously saved settings. Specific parameters can so be saved for each project.
3. Flagging only the second option: the same parameters are used for all the projects; the settings are saved in the registry keys and I cannot keep track of them in the .xml files as selecting the first option. The parameters are global and persistent inside Reconstructor®.
4. Flagging both options the sum of the effects is obtained: the parameters are global and persistent and you can keep track of their modification and use them as needed.

Cloud-to-cloud (and mesh) registration

The LineUp® tool includes a comprehensive suite of tools for coarse and fine registration of any amount of scans (and not only). The registration process is composed by several steps, depending on adopted survey techniques.

The general flow, here illustrated



starts from a preregistration step that allows you to compute a rough alignment between the imported models. The alignment can be later refined using ICP registration and Bundle Adjustment to choose and refine the good ICPs and discard the wrong ones, in order to reduce the global registration error.

See Getting Started -> Register to learn how to use the The LineUp® registration tools.

See Getting Started -> Georeference to learn how to use the The LineUp® georeferentiation tool.

Registration

- Automatic, target-free preregistration
- Automatic, target-based registration
- Manual pre-registration
- ICP registration
- Bundle Adjustment
- Scan alignment per groups

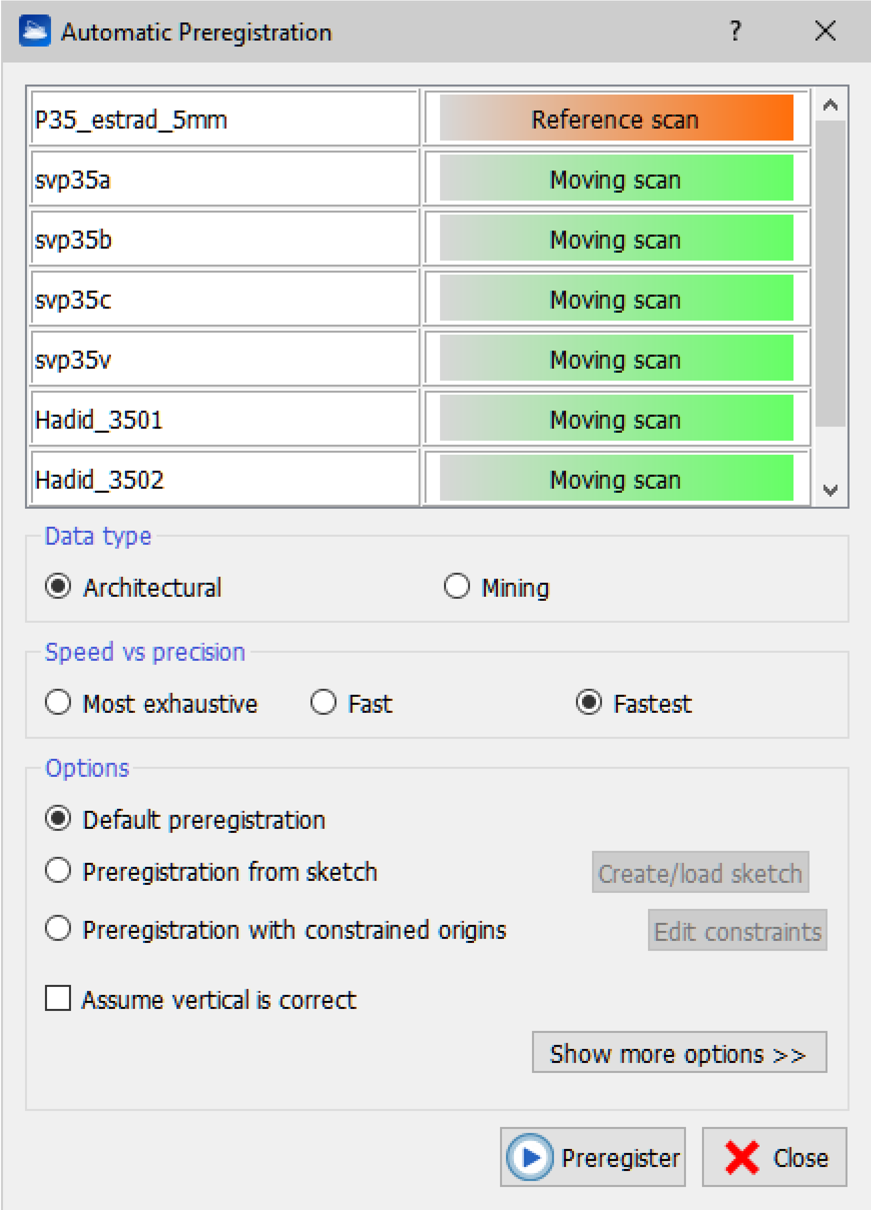
Georeferencing

- Point cloud georeferencing
- Manage control points
- Manage circular targets

Positioning

- Pose transform
- Determine the cloud's vertical direction
- Manual positioning (adjust Pose)

Automatic Target-free Pre-registration

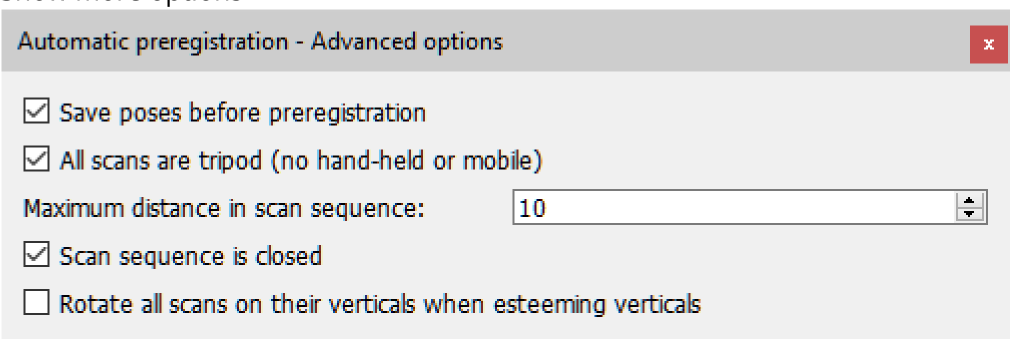


This dialog enables a very effective and fast procedure that preregisters a set of scans automatically, without using targets or markers.

This dialog enables a very effective and fast procedure that preregisters a set of scans automatically, without using targets or markers.

Here you can select the unique Reference scan that will be fixed.

Show more options >>



Features

- **No limits on the number of scans** to preregister. You can preregister 200 scans at the same time, the software will load at most two of them at a time.
- **Easy convergence with automatic search of best overlap.** To be registered together, the minimum overlap of two scans is 20-30% of their surface. However, in most cases you don't have to care about it, because of the following: given a set of N scans, the algorithm is

guaranteed to find all the scan pairs that match together, and to use the best overlapping N-1 scan pairs to preregister your dataset. You can customize this behavior via the "Maximum distance in scan sequence" option.

- **No point density requirements**. The algorithm adapts to the specific datasets, being it close-range indoor, or long range territorial data.
- **No complicated parameters to input** such as cell size or fitting strategy. You only specify if your dataset contains "construction" or "territorial" data: the algorithm will change accordingly the surface elements to look for and to match.
- **Very fast because of parallel computing**: the algorithm is guaranteed to do all computations in parallel, using all the cores your CPU has (unless you don't tell Windows to assign less cores to Reconstructor®).
- **Robust against symmetric datasets**. In case of rectangular empty rooms, stairs, or other symmetric datasets, Line Up Pro is guaranteed to quickly let you browse multiple preregistration solutions, and therefore to help you quickly find the right preregistration of your scans.
- **Easy to set constraints.** You can easily set constraints on the scan vertical. If you know that the scans are already on their vertical (with a confidence of 2-3°), you can tell the algorithm to leave the scans' vertical unchanged. If you don't, LineUp Pro will also esteem the scans' vertical, with the guarantee to correct angles between the given vertical and the correct one of max. 30°. As additional constraint, you can select which of the input scans is the reference scan during the preregistration.
- **It works for virtually all datasets**. This automatic preregistration technique has been tested with hundreds of different datasets containing from 10 to 200 scans per dataset. It gets your scans in a state of coarse registration without your intervention, or with very few easy corrections at the end.

Parameters

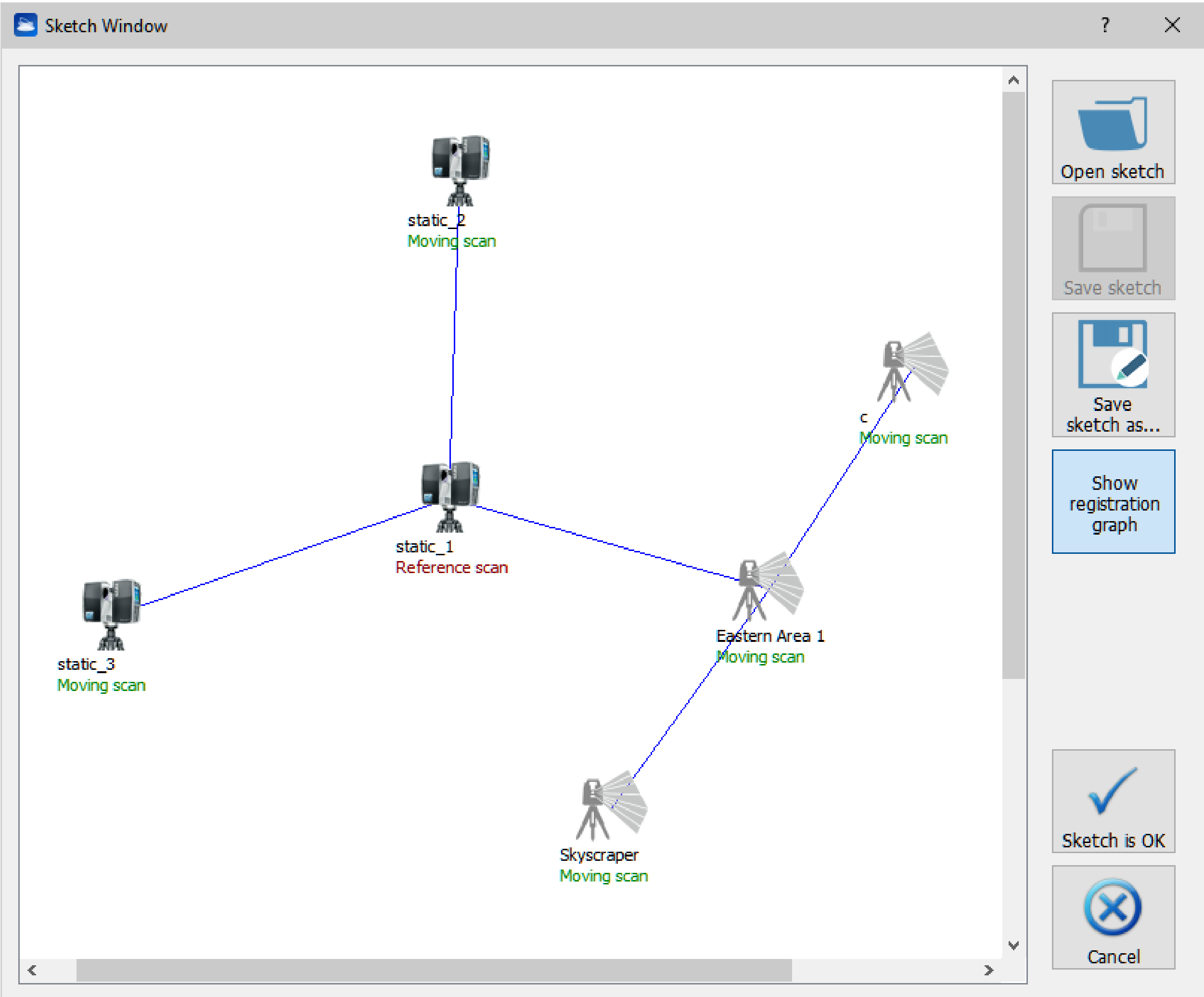
Here we give a top-to-bottom description of the parameters that you can set in the dialog.

- **Data Type:** here you can specify if your datasets relates more to a construction scene, with many planar surfaces, or if it relates more to a mining/terrain scene, with many irregular surfaces. This will greatly help the algorithm.
- **Speed vs precision:** this algorithm is designed to be fast in first place, and to deliver a solution in minutes or tens of minutes. We advice to leave this option to “Fastest” and to switch to the other two options only when the first does not work. The other two options indicate the algorithm to search for more matches between smaller surface elements. This leads in general to better results, but in some cases it can also spoil the results because it introduces more candidate matches for the final solution and therefore more ambiguities.
- **Preregistration from sketch.** By flagging this option, you tell the algorithm that you want to help it by creating a “sketch”, or a drawing with the approximate relative positions of the scans. To create or load a sketch, press the *Create/Load sketch* button. The Sketch Window opens for you to create the sketch.
- **Assume vertical is correct.** Some laser scanners feature an internal inclinometer, and therefore they are able to create raw scan data where the vertical is already correct, with an error of 2-3 degrees. If this is the case, then you can flag this option and the algorithm will match the scans without changing their initial vertical. If this is not the case, the the algorithm will also esteem the correct vertical before matching the scans. Please take into account that the maximum error in the vertical that can be corrected is 30 or 40 degrees.
- **Save poses before preregistration.** If you flag this option, Reconstructor® will automatically save the scans' poses in the Export/ project folder before starting the automatic preregistration. In this way, you will be able to easily go back to those poses

before the preregistration, with the function *Tools → Pose and registration → Pose transform → Apply pose from file.*

- **All scans are tripod (no hand-held or mobile).** Flag this command if you're using tripond laser scans
- **Maximum distance in scan sequence.** This parameter determines how many scan pairs the algorithm will try to match. It specifies the maximum distance in the scan sequence that two matchable scans may have, and it defaults to 10. For example: given an input sequence of 30 scans, Reconstructor® will try to match scan 5 with all the following [nearest], until scan 15. Scan 5 and scan 16 will not be matched because Reconstructor® will assume that they are too far away in 3D, being far away in the sequence. Generally speaking, given an input sequence of *N* scans and this parameter set to*M*, then Reconstructor® will try to match scan *s* only with the *M* preceeding and the *M* following it in the input sequence, considered as a circular sequence (scan*N* matches scan 1).

Sketch Window



This dialog enables a very effective and fast procedure that preregisters a set of scans automatically, without using targets or markers.

This window enables creation, editing and handling of *preregistration sketches*, created in Reconstructor®. Preregistration sketches are needed to describe the approximate relative positions of a set of scans, to give an initial hint to Line Up's automatic target-free preregistration. However, it is not compulsory to create a preregistration sketch to run the automatic target-free preregistration.

Usefulness

Creating or loading a preregistration sketch is useful mainly to speed up the automatic target-free preregistration : in fact, if the scans' approximate relative positions are known, the automatic preregistration algorithm will match only the scan pairs that belong to the *registration graph*: the blue polyline connecting the scans in the upper picture. Therefore, to preregister *N* scans the algorithm will match only *N* – 1 scan pairs instead of trying all the combinations, which can result in *N* (*N* – 1) maches. Therefore, automatic preregistration will be faster and will aim at the right solution more directly, avoiding matching the wrong scans with the subsequent ambiguity and confusion.

Another usefulness lies in keeping a sketch of your scans' positions. Yet another advantage of this window lies in the possibility of setting which scan is the reference scan, and optionally which scans should not be used during the registration.

How it works

In the main area of the sketch window, the sketch is shown. The scans are represented as icons on the sketch related to the scanner model. Below each icon, the scan name and the scan registration role are found. The registration role can be selected among three:

- **Reference scan**. The scan is not moved during the automatic preregistration.
- **Moving scan**. The scan will be preregistered against the others during the preregistration.
- **Scan not used.** The scan will not be preregistered. This is useful if you acquired scans that are not useful for the preregistration, like hi-resolution scans of small targets, or test scans.

Just drag the scan icons around the sketch to change their position. You can change the scans' registration roles via the *Edit registration roles* button on the left.

On the left, some buttons are present.

- **Open sketch, save sketch, save sketch as.** These buttons manage your sketch files, which have a ".sketch" suffix. You can create a preregistration sketch from scrath and then save it.
- **Show registration graph.** Toggle this button to show or hide the blue polyline that represents the best registration open sequence for your scans. Move the scans so that the registration graph is the best possible.
- **Edit registration roles.** Click this button to change the registration roles for each scan, as explained before.
- **Sketch is Ok.** Select this option to close the sketch window and to go back to the control dialog for starting the automatic preregistration taking as input the edited sketch.
- **Cancel.** Select this option to close the sketch window and go back to the automatic preregistration dialog without using the sketch.

Manual Pre-registration

- Manual Pre-registration (among grid point clouds)
- Manual Pre-registration (among models)

Manual Pre-registration (among grid point clouds)

The pre-registration technique allows you to manually compute a rough alignment between two grid point clouds. The alignment can be later refined automatically, using ICP registration.

Accessing the preregistration dialog

You can access this dialog from the button R*egistration* in Reconstructor®'s top toolbar.

Defining the reference and the moving grid

From the list of grids in the dialog, choose the reference model and press **Set reference grid**, then select the grid to move and press Set **moving grid**. The two grids appear as range images on the right and left panel of the dialog respectively.

Finding corresponding points

The preregistration procedure works by finding three couples of corresponding points among the reference and moving grids. If needed, you can register one moving grid to more reference grids: to do so, change the reference grid to another from the loaded ones, the image-grid point pairs are stored for each grid, and are accumulated.

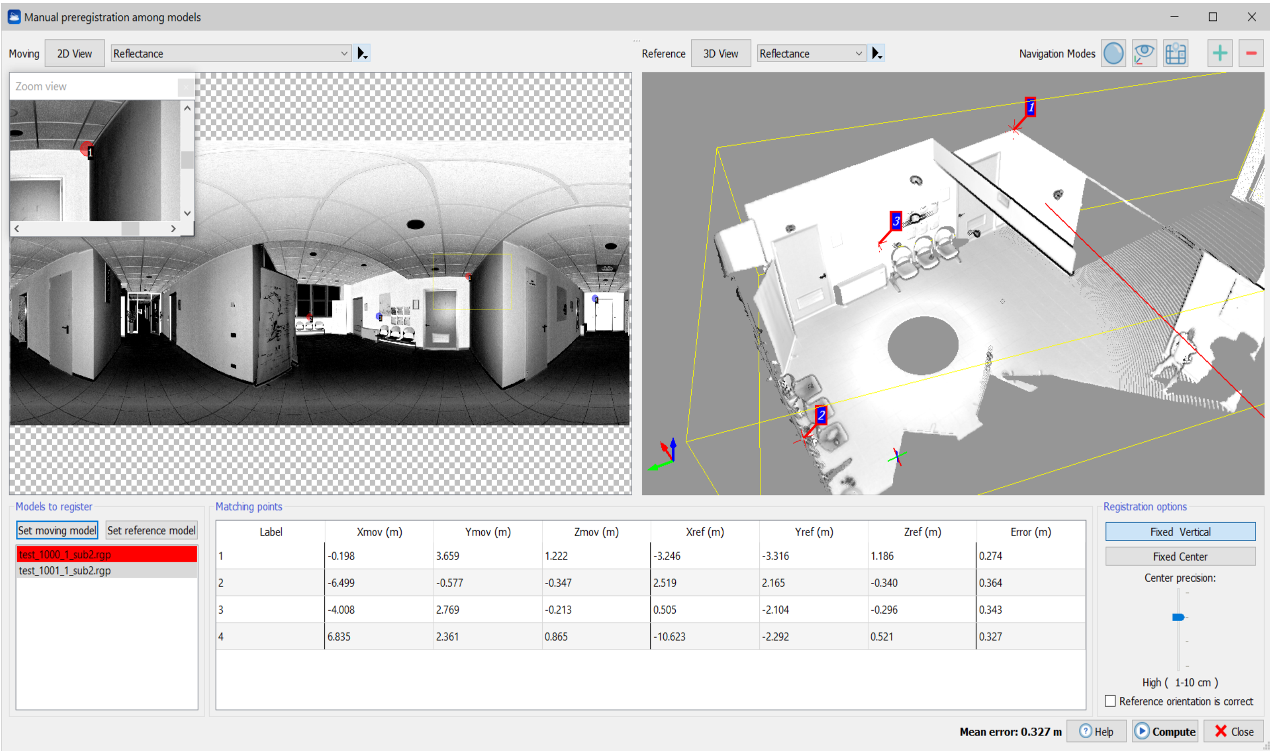
Keyboard/mouse event	*Effect*
LMB double click	Selects a point. Selected points are listed in the tables *moving grid* and *reference grid* at bottom of the dialog
Mouse wheel rotates	Zooms the range image in/out centering on the hovered point
Key 'z'	Toggles the Zoom views shown at the bottom of the screenshot
Ctrl + LMB + move mouse	Moves the zoomed region inside the global view
Space bar	Rotates the range image of 90°
Shift + LMB + move mouse	Translates the range image
RMB on a selected point	Deletes the point
LMB on a point + move mouse	Moves the point around
Alt + central mouse button	Fit image to window
Alt + RMB	Reset image to original size

When three or more couples of points are selected, the table *error per point* shows the alignment error per point couple. The mean registration error is instead shown at bottom left of the dialog. In general a pre-registration error of 2 cm is acceptable. However, this depends on the scale of your project and of the final output you want to get. If a point couple introduces too much error, you may try to move slightly around the points after zooming on them, to improve that couple's error.
Try to change the color type of the grid on top of the window to improve the contrast of features. Press the right arrow to show a menu of available commands for the current color type for better adjustment.
The list of points is shown in the table, where every cell is manually editable to force the values, like for geo-referentiation.

Finishing the preregistration

When at least 3 point correspondences are found the *Mean error* value appears in the bottom of the dialog. When the acceptable value is reached, press **Compute**. A dialog shows you the mean registration error and gives you the options to immediately apply the registration to the moving grid, to copy the registration transform to clipboard to apply it later, or to cancel if you don't accept the

error. You may want to copy the transform to the clipboard if you want to move more grids according to the same transform. This is useful for example if you have a group of say 10 grids already registered among them, to be registered against other 10 already registered among them.
To apply "later" a registration transform stored in the clipboard, select in the project window the grids you want to apply the transform to, right-click on them and select *Registration->matrix transform* from the context menu. Then select *Apply registration transform* in the *Matrix transform dialog.*
When the pre-registration is finished, the grids are unloaded from the project to free memory. Please reload them to see them rendered.



Manual Pre-registration (among models)

Manual preregistration

The manual preregistration can be performed not only between point clouds (structured or not) but also between mesh models and point clouds and models.

1. Select at least two models from Reconstructor®'s project view.
2. Select "Manual preregistration" among the registration tools. The dialog above appears.
3. In the list on the bottom right, select the *reference model* and the *moving model.* These two models are shown in the respective 3D views.
4. Navigate as usual in the 3D views, and
5. Find at least three pairs of corresponding points between the reference and the moving models.
   Double-click with the left mouse button on a model to select a point.
   Right-click with the mouse on a point to remove it.

6. Press the *Compute* button on bottom right, and select *Apply transform* if the registration error is satisfactory. The registration error doesn't need to be good, because it can be refined via the ICP registration.

Introduction

This function is part of LineUp's registration techniques. This pre-registration technique allows you to manually compute a rough alignment between two models (unstructured point clouds, structured point clouds, or triangle meshes). The alignment can be later refined automatically, using ICP registration. This function is useful in the rare cases when the Automatic preregistration function fails.

Reference and moving models

To use this function, please select from the project view at least two models. The models can be of any type: point clouds (structured or not) and/or triangle meshes.
When the preregistration dialog opens up, the selected models appear in the *Models to register* list on the bottom right. Please select a model as reference and a model to be preregistered against the reference. When you select a model either as reference or as moving, the model is rendered in the corresponding 3D window. Please use the buttons above each 3D window to switch between a *default view* and a *bubble view* of the model.
You can change the reference and moving models at any time.

Picking pairs of homologous points

To preregister the models, you indicate to Reconstructor® features that the models have in common, such as building edges, window corners, etc. **Double-click** with the mouse on a model to select a specific feature point. The point gets displayed in the 3D view with its label, and gets listed in one of the point tables on the bottom of the dialog, with its label and its cartesian coordinates shown in the current unit of measure. **Right-click** with the mouse on a point in the 3D scene to remove it from the list.
When you have at least three point correspondences, the *error table* on the bottom of the window displays the registration error associated with each point, to offer you a way to remove the worst point pair and select another one to improve the registration.

Finishing the preregistration

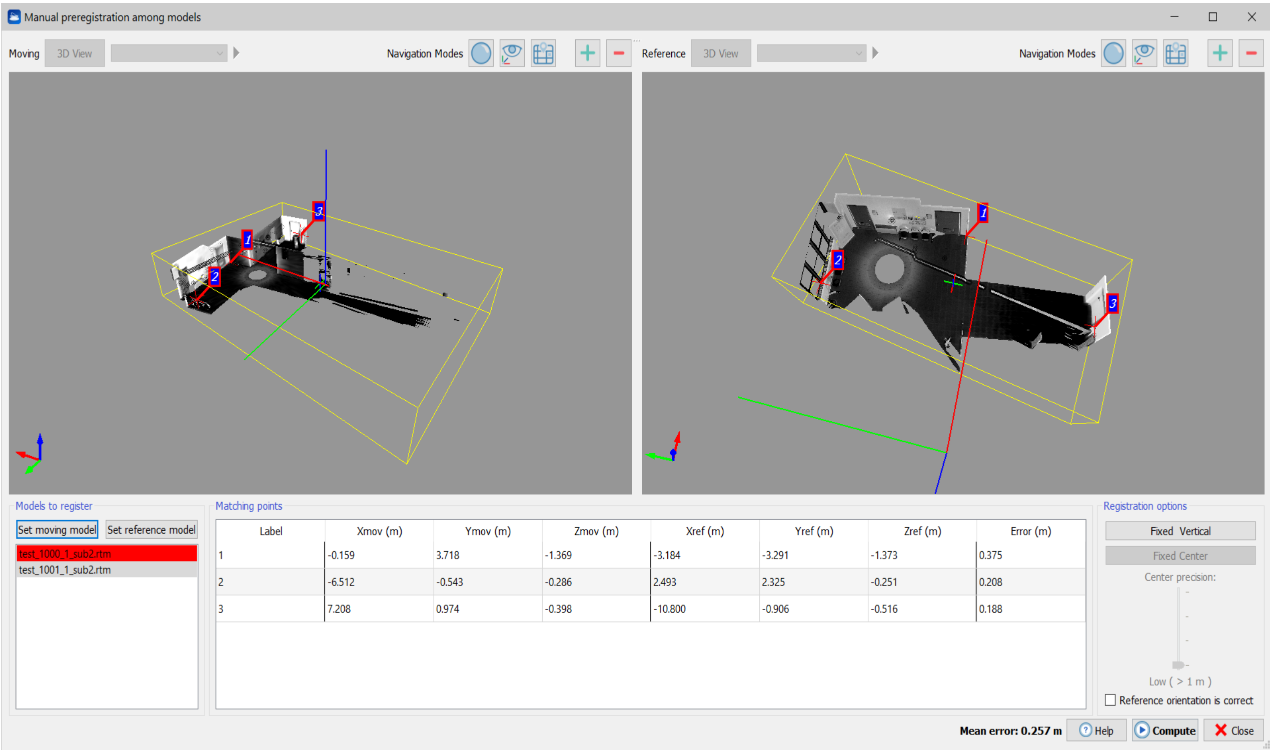
When at least 3 point correspondences are found press **Compute**. A dialog shows you the mean registration error and gives you the options to immediately apply the registration to the moving model, to copy the registration transform to clipboard to apply it later, or to cancel if you don't accept the error. You may want to copy the transform to the clipboard if you want to move more models according to the same transform. This is useful for example if you have a group of say 10 grids already registered among them, to be registered against other 10 already registered among them.
To apply “later” a registration transform stored in the clipboard, select in the project window the grids you want to apply the transform to, right-click on them and select *Registration->matrix transform* from the context menu. Then select *Apply registration transform* in the *Matrix transform dialog*.
When the pre-registration is finished, the models are unloaded from the project to free memory. Please reload them to see them aligned.

Registration constraints

On the bottom right of the dialog, you can also add some *registration constraints.* You can specify whether or not the vertical of the moving model is fixed. If you do so, you only need to specify at least two point correspondences instead of three to compute the preregistration. Additionally, you can specify the precision of the moving model origin's position, among four levels. If you specify that the moving model's origin is fixed, then you only need one pair of corresponding points to get a preregistration.

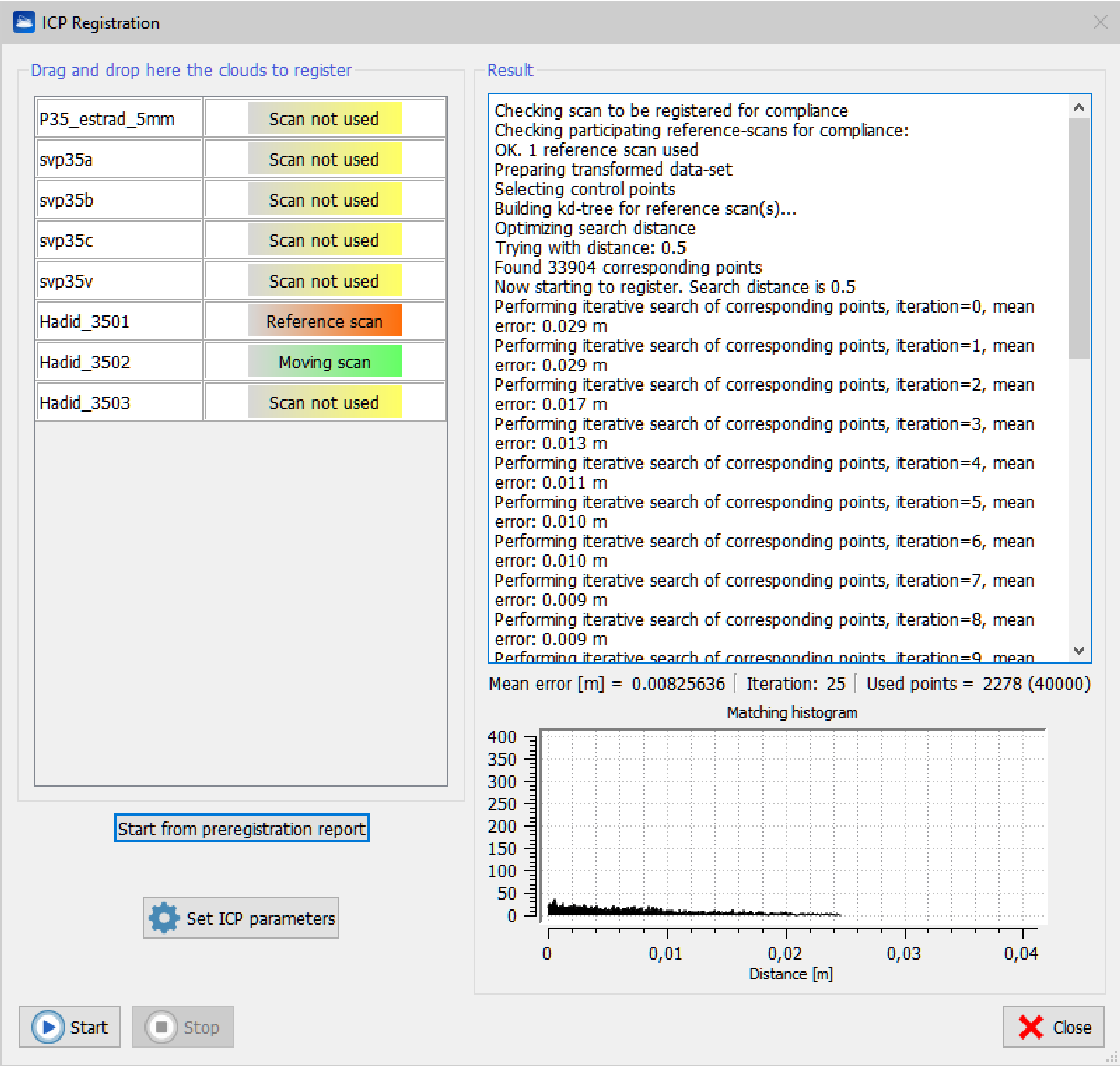
Computing mutual rotation among models

There are cases in which two models just need to be rotated around the vertical by some angle each, to get into a state of coarse registration. Many laser scanners embark GPS and inclinometers, therefore in some situation you may already possess the approximate values of the scans' vertical and origin. If that is the case, you can set the following registration constrains: *fixed vertical, fixed center* and de-flag the option *reference orientation is correct.* If you do so, you only need to select at least one pair of corresponding points, and Reconstructor® will compute the rotation angles around the vertical for both the reference and the moving model (the *reference* model will be moved as well).



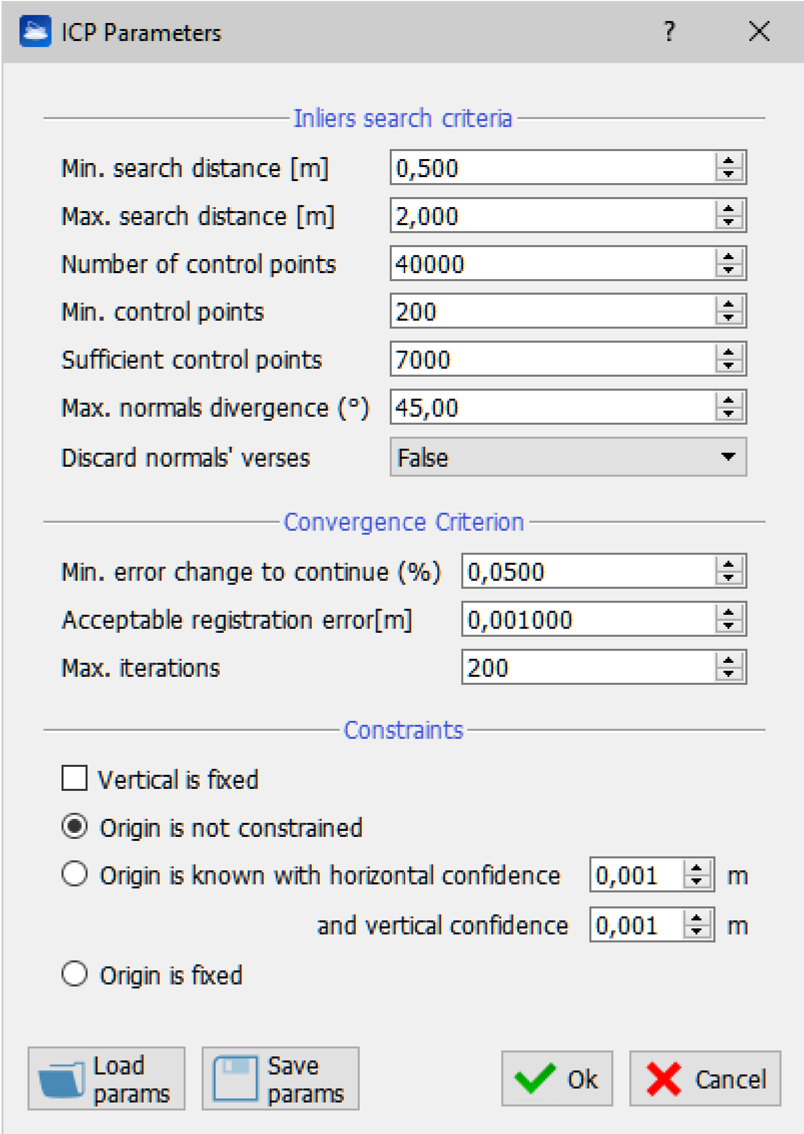
ICP Registration

ICP Registration is an algorithm to automatically perform fine registration of a moving point cloud against one or more reference clouds. The moving cloud must be roughly close to the reference cloud. It is possible to roughly align the moving cloud by **pre-registering** it.



ICP parameters

Below all the details.



Accessing the ICP Registration dialog

You can access this dialog from the button *Registration* in LineUp's top toolbar.
The ICP algorithm works on both structured and unstructured point clouds. The requirement is that the point clouds have the *normals* calculated, i.e. have the *Inclination* among their colors. If a point cloud doesn't have the inclination, during processing ICP will issue an error message. Inclination can be computed for grid point clouds during preprocessing.
Just drag and drop point clouds from the project window to the ICP dialog to register them. The clouds will appear in the top-left list in the dialog.

Setting the clouds' registration roles

In the top-left list, each cloud is listed with its name and its *registration role*. By clicking the button at the right of any cloud's name, you can select its *registration role* among three: each cloud can in fact be a *reference scan*, or a *moving scan*, or simply *not used*.
A reference scan will remain still in its position during registration, while a moving scan will be moved to align it better to the reference clouds. You can have as many reference scans as you want, but you must have exactly one moving scan to run ICP.
Example: suppose you need to register 10 clouds together. You may want to drag and drop them all onto the ICP dialog, then register the first two, then register the third using the first two as

reference, and so on, gradually registering the point clouds on the ones already registered.

Using a preregistration report to finely register a set of clouds

If you have a set of clouds already preregistered using automatic preregistration, you can automatically run in batch a sequence of ICPs that will register your clouds. Press the button *Start from preregistration report* , then select the preregistration report you want to use (usually found in the *Results/Reports* project folder). Below the button, a confirmation message appears ensuring that the report has been correctly loaded and that you can press *Process* to register all your clouds. After that you have loaded the preregistration report, you are not allowed anymore to change the registration roles, since the reference and moving scans are already defined by the report. This technique will use the minimum number of pairwise ICPs to register your clouds: if you have *N* clouds, *N* – 1 ICPs will be needed, as many as the edges of the *minimum spanning tree* that connects all your clouds. The information about which is the minimum spanning tree is contained in the preregistration report.

Comparison with Bundle Adjustment. Reconstructor® offers two algorithms to finely register a set of point cloud starting from a preregistration report: the one usable via this dialog, and *Bundle Adjustment*. Which one to use?

As said, the algorithm available here uses the minimum possible number of ICPs to connect your clouds to each other. Therefore, if your clouds form closed sequence (e.g. a loop around a building, or inside a hall), you have no guarantees that the first cloud will be aligned with the last. Bundle Adjustment, instead, is designed to use *as many ICPs* as possible, to connect each scan with all its neighbours in order to globally register all your data.

In conclusion: if your scans are positioned in a strict sequence, you should use the present method, because bundle adjustment may try to register together clouds that have no significant overlap with each other. If, on the other hand, your scans form a loop or a closed sequence, you should definitely avoid the present method and use a bundle adjustment.

Setting the ICP parameters

ICP stands for *Iterative Closest Point*. The algorithm finds points on the moving cloud that are close to the reference clouds. These points are called *control points* or *inliers*. The algorithm then iteratively moves the moving cloud to reduce the distance of the control points to the reference models. After each step of movement, the control points are recomputed.

Once the reference and moving clouds are set, the user can also tune several parameters of the ICP algorithm, displayed in the *Parameters* box on top right of the dialog. The default parameters should work fine for most cases. However, here a full explanation of the ICP parameters is provided.

ICP parameters are divided in three groups: parameters that define how to *search for inliers*, parameters that define the *convergence criterion*, and parameters that define *registration constraints.*

Inliers search parameters

- **Max search distance.** If the models have a bad pre-registration, try to increase the Max search distance, but the process will be slower.
- **Number of control points.** Tune the number of control points to the size of the models. The model to register (not the reference) is sampled with *Number of control points.* At the beginning of ICP, a *search distance S* is computed. Control points from the moving cloud are considered only if they are closer to the reference models than *S*. *S* is calculated so that it is between*Minimum search distance* and *Maximum search distance*, and so that at least *Sufficient control points* can be found in the moving cloud to be closer than *S* to the reference clouds. If such an *S* cannot be found, for example because the clouds are too far away and therefore the control points are too few, ICP stops and an error message is issued.

- **Sufficient control points.** See explanation above.
- **Minimum search distance.** See explanation above
- **Min number of control points.** During ICP iterations, the control points are recalculated. If at a certain iteration they are fewer than this value, ICP stops and an error message is issued.
- **Max normal divergence [deg].** Two points close to each other are considered not matching if their normals are diverging more than this angle. This is useful to filter out from the registration noisy data like vegetation, foliage, or moving objects like cars or people. The assumption is that points belonging to the same object should have also the same normal direction.
- **Min. scan bounding box overlap (%).** At the algorithm's beginning, the moving cloud's bounding box is compared with the bounding box of each one of the reference clouds. If the overlap between the bounding boxes is lower than this value, the reference cloud is not considered by the algorithm.

Convergence criterion parameters

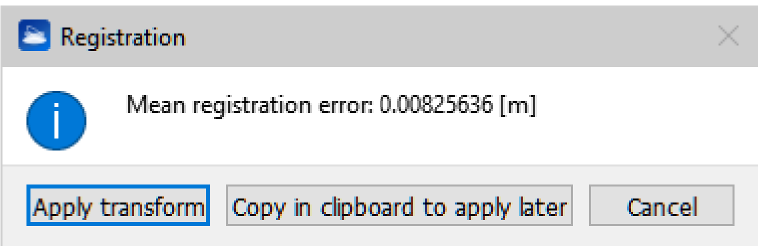
- **Registration error [m].** Tune this parameter to select the desired registration accuracy. ICP will stop iterating if the mean registration error goes below this value. The other parameters that influence ICP's *stop criterion* are: *Registration delta-change* and *Maximum number of iterations*.
- **Registration delta-change (%).** ICP stops if the error-delta between three iterations is smaller than this parameter. Let's call this parameter *d*. Let's also define *e*(*n*) to be the mean registration error at iteration *n*. ICP will stop iterating if, at iteration *n*, $[(e(n-3) - e(n)) / e(n)] < d$.
- **Minimum number of iterations.** ICP will execute at least this number of iterations, and then will start checking the break criterion.
- **Maximum number of iterations.** ICP will do at most this much of iterations.

ICP constraints

- **Vertical is fixed**. Set it to true if you know that the moving scan is already set on its vertical in the current UCS.
- **Translation is fixed.** Set it to true if you want to constrain the origin of the moving cloud to be fixed. Ths is useful if you know with great precision (e.g. via total station) the coordinates of the scan's origin.

When the parameters are set, press the button *Process* on bottom left. ICP begins, showing log messages in the right-central *Result* panel of the dialog. Also, the graph on bottom right displays the error histogram for each iteration. This is useful to see at a glance whether the algorithm is converging.

Finishing the registration

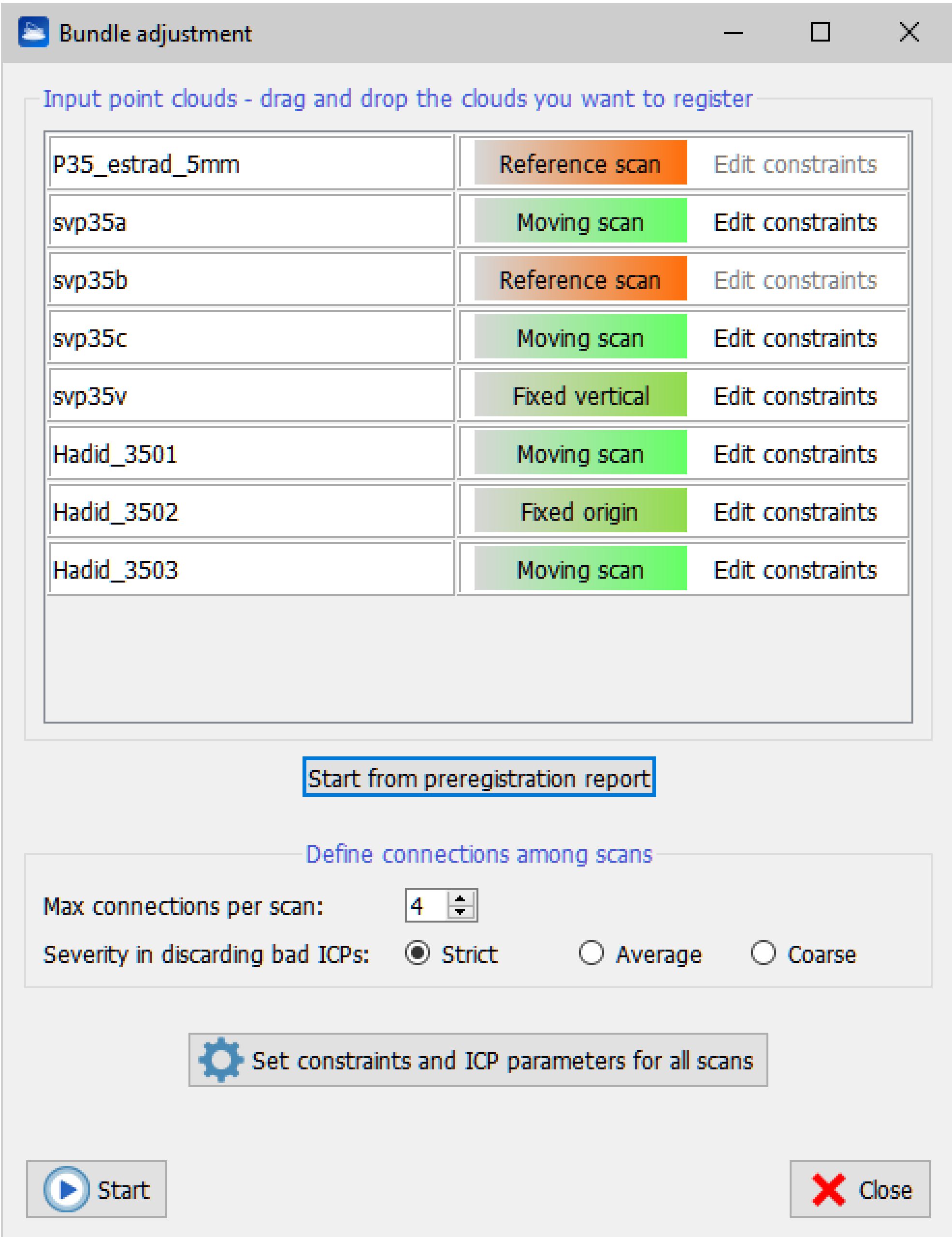


After finishing the registration, the dialog pictured above shows you the mean registration error and offers you three options: to immediately apply the registration to the moving grid, to copy the registration transform to clipboard to apply it later, or to cancel if you don't accept the error. You

may want to copy the transform to the clipboard if you want to move more grids according to the same transform. This is useful for example if you have a group of say 10 grids already registered among them, to be registered against other 10 already registered among them.
To apply "later" a registration transform stored in the clipboard, select in the project window the grids you want to apply the transform to, right-click on them and select *Registration->matrix transform* from the context menu. Then select *Apply registration transform* in the *Matrix transform dialog*.
When the registration is finished, the grids are unloaded from the project to free memory. Please reload them to see them rendered.

See Scan alignment per groups.

Bundle Adjustment



Solving a puzzle of a thousand pieces

To understand bundle adjustment, we can imagine to solve a big puzzle with many pieces. Each

piece matches with four others, sometimes the match is easy, sometimes you have to slightly push a piece inside another, and sometimes two pieces just don't match. Often, you need to know the neighboring pieces to decide whether two pieces match correctly or not. And the pieces reach their final, adjusted positions only when the puzzle is complete. If ICP registration can be seen as finding the match between two pieces, Bundle Adjustment means completing the whole puzzle by choosing and refining the good ICPs and discarding the wrong ones.
This algorithm allows to register together many point clouds, distributing evenly the registration error. The user specifies which point clouds are *reference* clouds (they are locked during the registration) and which are *moving*; the moving clouds will move and align on the reference clouds and between them during alignment. While the ICP registration works only for one pair of clouds at a time, this algorithm registers together *N* clouds at the same time, diffusing and minimizing the global registration error. The input point clouds must be *pre-registered*. They do not need to be structured but they need to have the normals. There is no upper limit to the number of the input clouds.

How does it work

Given a set of *N* scans, the algorithm will create as many *connections* among pairs of neighboring scans as possible. For each connection, a pairwise ICP will be executed, using the Settings accessible from the Line Up top toolbar. From each ICP, the best matching point pairs between the two scans will be saved. In the end, a final nonlinear minimization step will be run only among these matching point pairs of all the connections. The global registration error among these point pairs is minimized, having as unknown variables the scans'poses.

Parameters

Input point clouds

Just drag the poing clouds you want to finely register from the project view, and drop them on the Bundle Adjustment dialog. They will appear in the top list. On the right of each cloud, a button appears that can be toggled among red (reference scan) and green (moving scan). Therefore here you can decide which scans must remain still during registration and which must be registered.

Preregistration report

If you have previously performed the automatic preregistration of your scans, you can use the results of your preregistration in the bundle adjustment. Click ***Start from preregistration report*** and load the report of the automatic preregistration, a text file by default saved in the /Exports folder of your project. The dialog will show in the top list all the clouds that were automatically preregistered, and will adjust the ICP settings accordingly to the preregistration error contained in the report.

Defining connections among scans

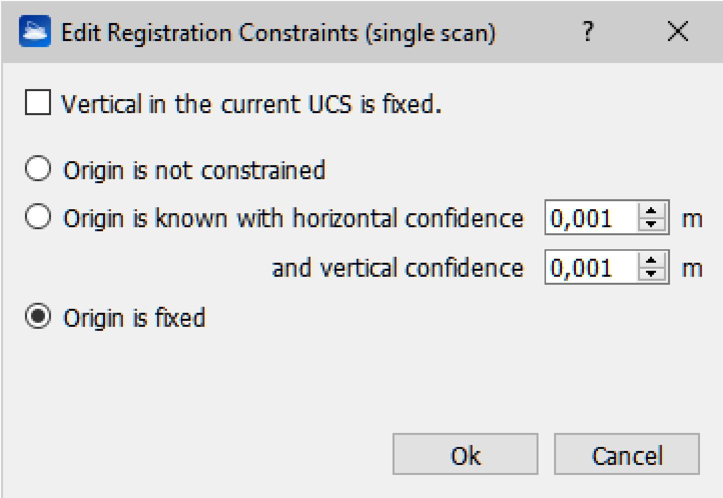
To reach the best global registration possible, this algorithm will use as many *connections* as possible among your scans. Knowing the topology of your scans, you can set the ***Max connections per scan*** parameter accordingly. If you scanned the exterior of a building, then it may be that each scan shares geometry with 3 or 4 neighbors. If, instead, you scanned a room or a hall, then it makes sense to say that each scan matches with other 5 or 6 scans, therefore all pairwise ICPs between a given scan and its 5 or 6 neighbors should be launched. However, doing so there is the risk to run ICP between two scans that have no geometry in common. In this case, the ICP error will be very high.Therefore, the algorithm will discard bad ICPs and keep the good ones. You can set the ***Severity in discarding bad ICPs*** among three levels as in the dialog. We advice to always select “Strict”, and only if your data are particularly noisy (e.g. mining data) then select “Coarse”.

Global constraints

You can specify some constraints that must remain valid for all the scans during all the registration. Below in the dialog, you can flag that ***all scans have fixed vertical*** . This is useful for example if you have imported scans from a scanner with vertical corrector and you trust the vertical to be correct below 1°. You can also flag that ***all scans have fixed origin.*** This means that you know already the coordinates of all the scans'origin with a precision higher than the point density (~ expected registration error), therefore the scans will only be allowed to rotate during bundle adjustment.

Scan-specific constraints

The top of the window shows the list of scans to be registered. On the right of each scan, a button allows you to switch between whether the scan should be *reference* or *moving*.



When the scan is moving, still you can define scan-specific constraints via the ***Edit constraints*** button. Clicking this button, a dialog appears where you can flag whether the scan has the vertical fixed or not. Moreover, you can constrain the scan's origin in three levels. The origin can be free to move (not constrained), can be constrained within a certain horizontal and vertical interval, and can be held fixed.

Output

Once you have defined the connections among the scans and the additional constraints, press *Start* to begin processing. A progress bar appears, and you can also open the log window to have more detail on what's happening.

At the end, the clouds will be moved to the new positions and their files will be saved. Therefore, you may want to save the clouds' poses before running bundle adjustment. Moreover, a dialog appears indicating the number of registered clouds and the global registration error. After you press OK, you can click on ***Save PDF report*** to get a much more detailed PDF report. There also a ***Save .txt report*** option to save a text version of the full report that can be copied to Word or Excel.

Resolution of common problems

I have run bundle adjustment, but scan A didn't match well with scan B

Get the report of the bundle adjustment, and check if the edge between A and B (or B and A) is listed in the section "Report by edge, sorted by error". If it is, most likely the ICP between A and B has converged bad: you can re-run the ICP only between A and B with different parameters, and then re-launch the bundle adjustment having A and B among the references.

However, most likely the edge between A and B will be in the "Edges not used" section of the report, or even not in the report at all. If the edge (A, B) is among the not used, then try to re-launch the bundle adjustment decreasing ***Severity in discarding bad ICPs.*** If the edge (A, B) is not in the report, then try to increse the ***Max connections per scan*** and re-launch the bundle adjustment.

I have run bundle adjustment, but I see that the scans did not converge well, moreover the global registration error is 4 cm while it should go down to 2 cm.

Press the *Settings* button of the Line Up top toolbar, and go to the ICP settings. Set the ICP parameters so that only the very best inliers are selected. You can lower the min search distance to 0,2 m or to 0,1 m, lower the sufficient control points to 3000, and lower the max divergence among normals to 10° or even down to 3°. Then, re-launch the bundle adjustment. Only the best matching points among scans will be selected, and the error among them will be minimized, resulting in a better convergence and tighter error. You can repeat the process to improve the results.

See Bundle Adjustment Report.

Bundle Adjustment Report

At the end of the bundle adjustment process you can save a PDF (or txt) report detailing the ICP and Bundle Adjustment results.
Here are explained all the voices inside this report.

Summary

[N] scans have been finely registered with bundle adjustment.
Num ICPs used: number of ICPs actually used
Average ICP error [m]: average value of registration errors related to all scan pairs
Average registration error of corresponding points [m]: global registration error reached between the most important corresponding points. This error is the main result of the bundle adjustment.

Input parameters

In this section the parameters that you set before the bundle adjustment are summarized:
Max neighbors for each scan [N]: number of scans (close to the current scan) taken in account
ICP selection severity: Level of severity used in discarding bad ICPs
ICP min search distance [m]: see Inliers Search Parameters in Cloud to Cloud Registration set parameters
ICP max divergence among point normals [deg]: see Inliers Search Parameters in Cloud to Cloud Registration set parameters
List of registered clouds: the well registered point clouds are here listed

Detailed results

ICPs executed: number of ICP processes that have been executed during bundle adjustment.
ICPs used: number of ICP registrations that have been found good enough to contribute to the final bundle adjustment.
Average ICP error: average value of registration errors related to all pairs of scans
Maximum ICP error allowed [m]: see Inliers Search Parameters in Cloud to Cloud Registration set parameters
Total cost before minimization [m]: statistical parameter derived from least squares method, relative to state before minimization
Total cost after minimization [m]: statistical parameter derived from least squares method, related to state after minimization
If Total cost after minimization < Total cost before minimization an error decrease is achieved
Reason of convergence: if the value of |cost_change|/cost is lower than 1.000000e-008 (fixed upper limit) the function tolerance is reached.
Average registration error of corresponding points [m]: global registration error reached between the most important corresponding points. This error is the main result of the bundle adjustment.
N° corresponding points: these are the most important corresponding points among all valid scan pairs. The final global error minimization is run exactly on all these corresponding points.
N° iterations: number of iterations in the iterative decreasing error process
N° parameters, **N° effective parameters**: these are the parameters of the error functional to minimize, and depend on the degrees of freedom of the problem. If bundle adjustment is run among ten scans for example, keeping one as reference, then the effective parameters are 9 moving scans * 6 degrees of freedom (three for translation and three for rotation) = 54. However, if you change the registration constraints (e.g. adding more reference scans, imposing fixed vertical and/or fixed translation) the number of effective parameters changes accordingly.
N° threads used: Bundle adjustment uses as many threads as the logical CUP cores of your PC.

Therefore, bundle adjustment is very CPU-greedy to offer you the fastest processing time. If you desire another behavior, you can assign less processor to Reconstructor from the Windows task manager.
Time taken for final bundle adjustment [mm:ss.msec]
Total elapsed time [mm:ss.msec]

Report by edge, sorted by error

All the analyzed couples of scans (i.e. edge A and B) are sorted regarding the registration errors at the end of ICP and bundle adjustment phases.
[N] corresponding points found between scans *A* and *B*
ICP error [m]: registration error related to single pair of scans, as they are isolated from others.
Correction after bundle adjustment [m]: by how much the registration error related to the single scan pair has changed after the global error minimization.

Edges not used

If all scans are finely registered *All edges have been used* is written here, otherwise a list of discarded edges, sorted by error, is shown. If you see among the “edges not used” a scan pair that should be registered together according to you, then you may want to re-run Bundle Adjustment with coarser parameters. In the same way, if a scan pair has been registered together that you think it shouldn’t have, you can re-launch Bundle Adjustment with stricter parameters.

Scan alignment per groups

What you can do if the automatic pre-registration fails?
Here the instance of a set of scans (and/or 3D models) that cannot be aligned because of a low overlap between some scans.
If it is possible to detect groups of scans already registered and one scan for group that has a good overlap each other, a manual pre-registration can be run.

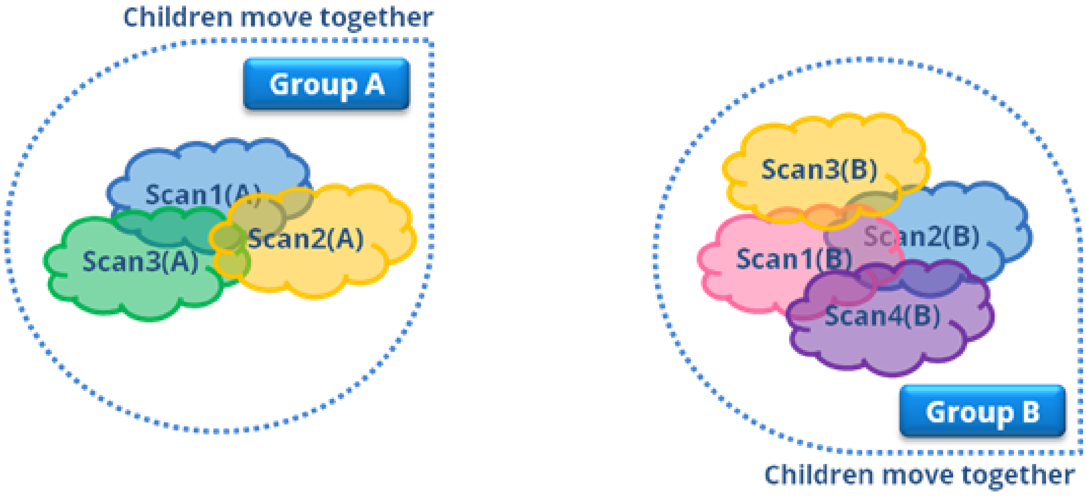
A *group* is a cluster of items composing the project.
A Registration role for all the scans inside the group can be set: *None*, *Same as parent*, *Children move together.* These roles are used to carry out a registration through groups.

The workflow is the following. Letting us help by an example...

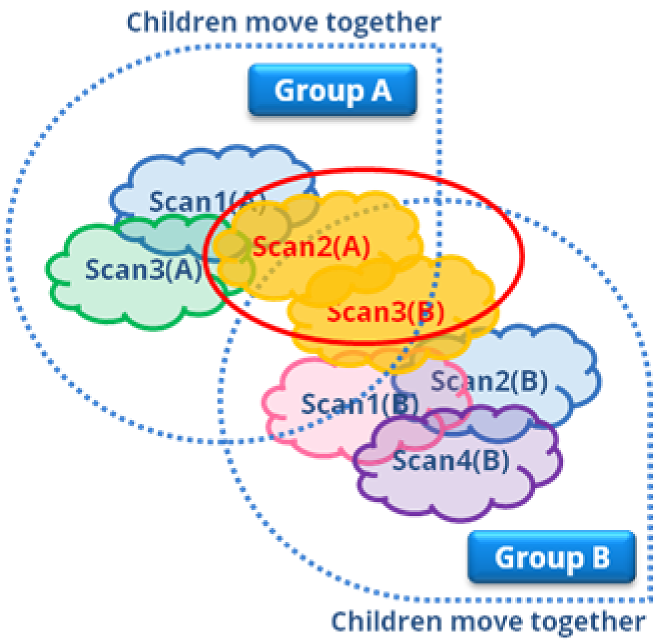
1. Find the groups of scans already registered.
   There are two possibilities to create groups:
   a. Automatically
      When you automatically register scans with the LineUp® tool, groups containing just registered scans are created by default
   b. Manually
      you can manually create a group already registered scans with the specific command.

   In the example below, we indentified two groups: A and B.
   We want to move all the scans of the Group B onto the scans of the Group A, without losing the alignment of the scans previously done.

2. Apply the “*Children move together*” registration role to the groups that you want to align. If a single scan is moved in the 3D space, all the other scans move together, as a rigid system.



3. Find, for each group, a scan with a good overlap with one or more scans in other groups [in the example Scan2(A) and Scan3(B)].
4. Carry out a manual pre-registration among models to align Scan2(A) and Scan3(B).



Since the "*Children move together*" registration role of the groups permits a rigid and common movement of all the scans of the groups, this procedure is enough to align all the scans.

Target-based Registration

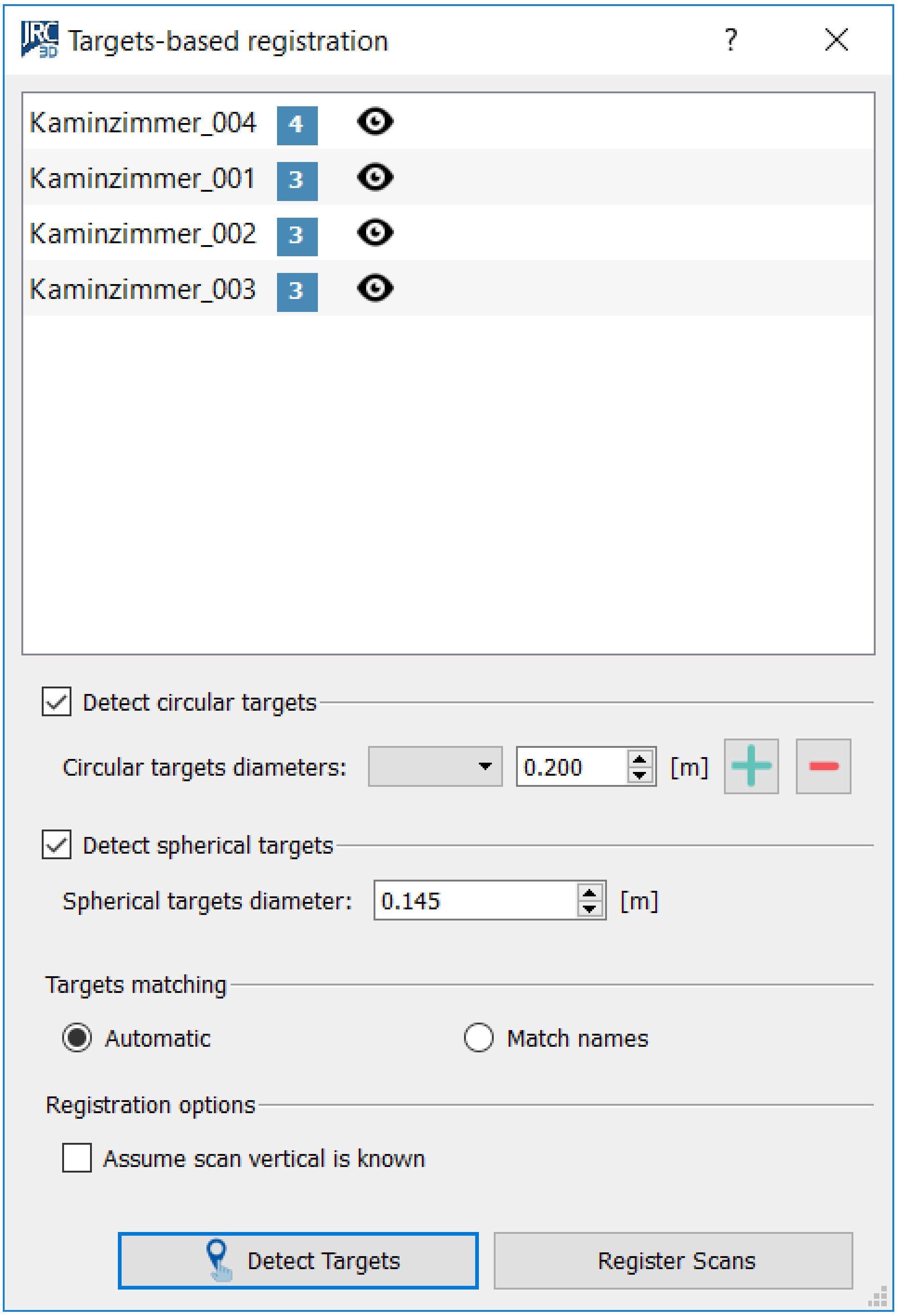
In some cases, especially when the geometry of the environments does not have many features, it is useful to use the automatic pre-registration or registration with targets.

Reconstructor® can manage:

- Planar Circular Targets;
- Real recognizable Points.

Manage them in Targets Editor window.

Target registration



This function automatically registers a set of scans by detecting and matching targets. The user can start the automatic target detection, otherwise can manually add, move and remove targets.

For each selected scan, the number of detected targets is listed. By clicking the *eye* icon, the user can open the Target editor dialog for the selected scan.

The *Detect Targets* button allows the user to perform the target finding stage on the selected scans. The targets associated with the selected scan are overwritten.
The *Register Scans* button performs the target registration stage. To register scans using targets, at

least three non-collinear targets are needed for each scan. Two targets per scan are needed if the *Assume scan vertical is known* checkbox is checked.

Via the checkboxes, the user can select wheter to detect circular flat targets. If*Detect circular targets* is checked, the user can add one or more diameter values by typing the numeric value and pressing the “plus” button. An inserted diameter can be removed by selecting it from the list and clicking the “minus” button.

See Manage control points and Manage circular targets and for details.

Alignment Options

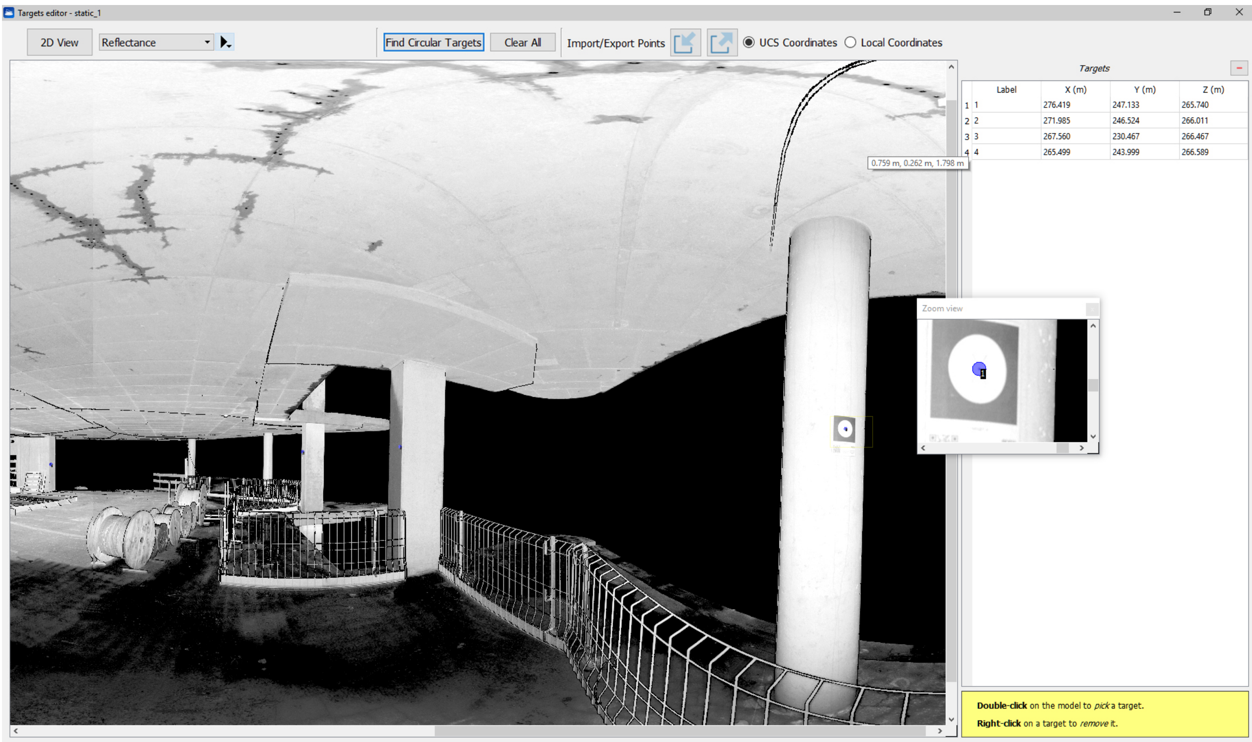
The user has two matching options to choose from:

- Automatic: the scan will be registered along the best of all possible matches among the targets
- Match names: the targets with equal labels will be matched together. This option is useful when the user is sure that same targets in different scans have the same label.

The user can also select whether the scan vertical in the current UCS must be used as a constraint to registration (and therefore left unchanged) or whether the system can also move the scans vertical to get the best registration.

If a fair amount of targets is used and the Automatic Target-based registration process is successfull, the Bundle Adjustment registration is not mandatory.

Targets Editor



The targets editor dialog shows a 2D view of the current scan with its associated target points. The user can manually add, move and remove targets.

Additionally the user can also start the automatic target detection by clicking the *Find circular targets* button to automatically detect planar circular target of given diameters

Mouse controls

- double click on the model to add a generic target point;
- right click to remove an existing target;
- drag the cursor while pressing the left mouse button to move an existing target.

On the right side of the window the current targets with their 3D coordinates and confidence values are listed. The zoom window can be centered on a specific target by clicking on a target from the list. A target's label can be edited by double clicking the respective cell in the targets list. Note that a target label must be unique among the same point cloud.

You can find targets both in 2D and in 3D view and with different color layer.

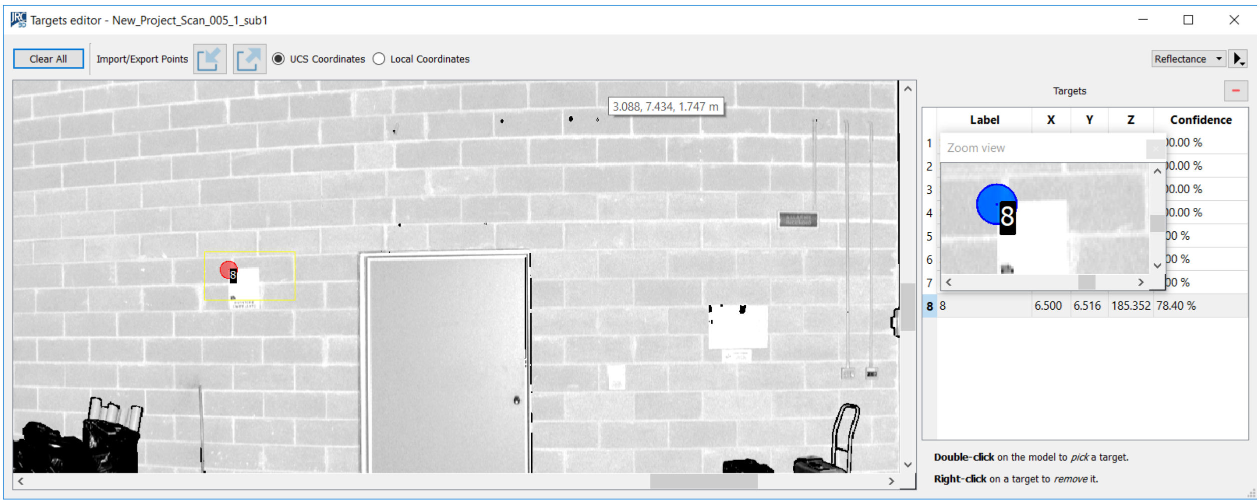
The *Clear All* button permits to delete all targets previously found in the scan.

It's also possible to *Export* and *Import points* to Text files (both in UCS and in Local coordinates) through the relative buttons.

A customized Target Editor will be open if Manage control points and Manage circular targets (in Georeference tool) are selected.

Manage Control Points

This function opens the Targets editor dialog, focused on control points that aren't circular targets, so common points in the point cloud



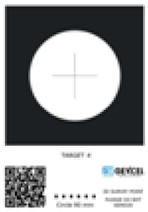
The targets editor dialog shows a 2D view of the current scan with its associated target points. The user can manually add, move and remove targets.

Mouse controls

- double click on the model to add a generic target point;
- right click to remove an existing target;
- drag the cursor while pressing the left mouse button to move an existing target.

Manage Circular Target

This function opens the Targets editor dialog, focused on planar circular targets recognition



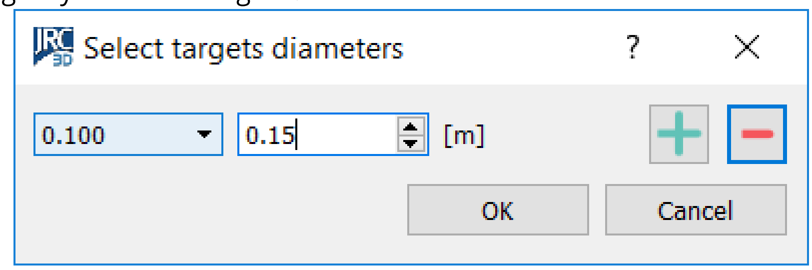
A circular target is a planar target printed as a white circle against a dark background. The value of internal diameter of the circle has not limitations.

Automatic detection of circular targets

Reconstructor® supports an easy-to-use and fast automatic circular targets detection algorithm for tripod scans. It needs only one input parameter (the diameter of the required circle). Several diameters are searchable at the same time.

Go in *Line Up,* selecting *Georeference* Tools and *Manage circular targets*, and selecting a grid point cloud.
A dialog appears showing a 2D view of the structured cloud.

Click *Find Circular Targets* on the top left. A small dialog appears where you can input the diameters of the circular targets you are looking for.



You can add one or more diameter values by typing the numeric value and pressing the “plus” button. An inserted diameter can be removed by selecting it from the list and clicking the “minus” button.

After inserting the diameters and clicking *OK* button, Reconstructor® automatically finds all circular targets in the given point cloud, and displays the targets list on the right of the dialog, populated with the found circular targets. Just click on any target on the list and you will see the corresponding one in the *Zoom View*: this is useful to check whether any false positive has been detected.

Mouse controls

- double click on the model to add a generic target point;
- right click to remove an existing target;
- drag the cursor while pressing the left mouse button to move an existing target.

Conditions and limitations

To recognize a circular target, the resolution of the scan should be such that at least 5 points can be detected on the circumference (on the edge between white and dark areas).

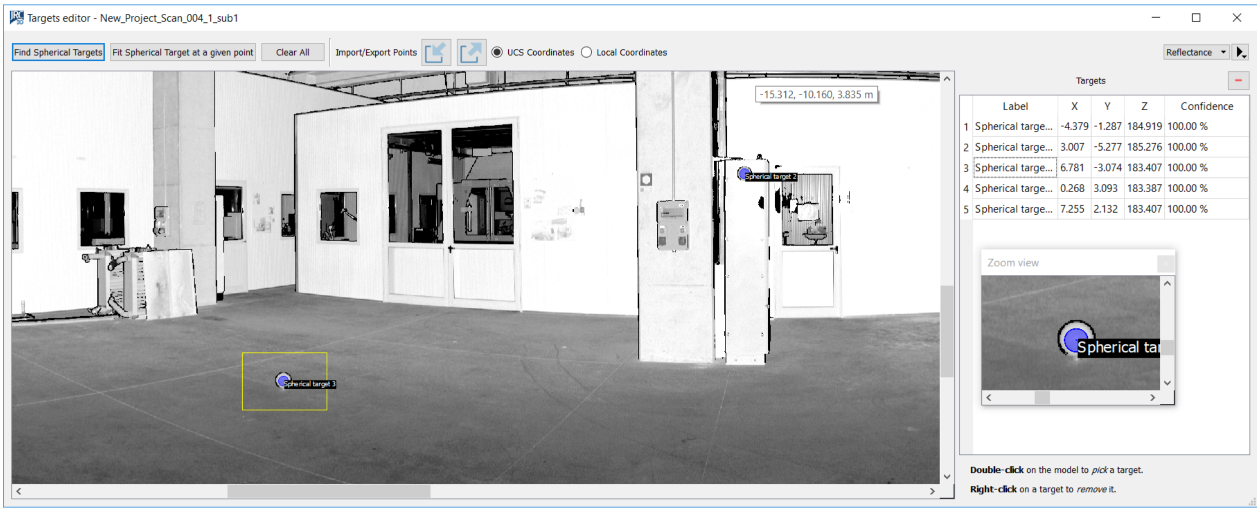
Manage Spherical Targets

This function opens a customized Targets editor dialog, focused on spherical targets

Automatic detection of spherical targets

To get quality surveys, scanner operators often place spherical targets on the field; these spherical objects can be automatically detected and used therefore as precise targets for registration. Reconstructor® supports an easy-to-use and fast automatic spherical targets detection algorithm for tripod scans.
It needs only one input parameter (the diameter of the required spheres) and it is robust to errors such as false positives or missed spherical targets.

It is possible to automatically detect spherical targets by going in *Line Up,* selecting *Target-based registration* tools and *Manage spherical targets*, and selecting a grid point cloud.
A dialog appears showing a 2D view of the structured cloud. Click *Find Spherical Targets* on the top left. A small dialog appears where you can input the diameter of the spherical targets you are looking for. The default value is 14.5 cm. After inserting the diameter, Reconstructor® automatically finds all spherical targets in the given point cloud, and displays a dialog with the number of found sphere, the average sphere fit error (the average distance of the inlier points to their own sphere) and the elapsed time. After clicking *Ok*, the targets list on the right of the dialog gets populated with the found spherical targets. Just click on any target on the list and you will see the corresponding sphere in the *Zoom View*: this is useful to check whether any false positive has been detected.



Accessibility and workflow

It is also possible to detect the spherical targets for a set of grid point clouds, by selecting them, then selecting *Pose & registration* and *Targets registration*. By doing so, the Target-based registration dialog appears. Uncheck the *Detect circular targets* checkbox, leave *Detect spherical targets* checked, select the desired spherical targets diameter, and click on *Detect targets* on bottom left. The spherical targets will be automatically detected for all the listed clouds, erasing any target previously belonging to the clouds.

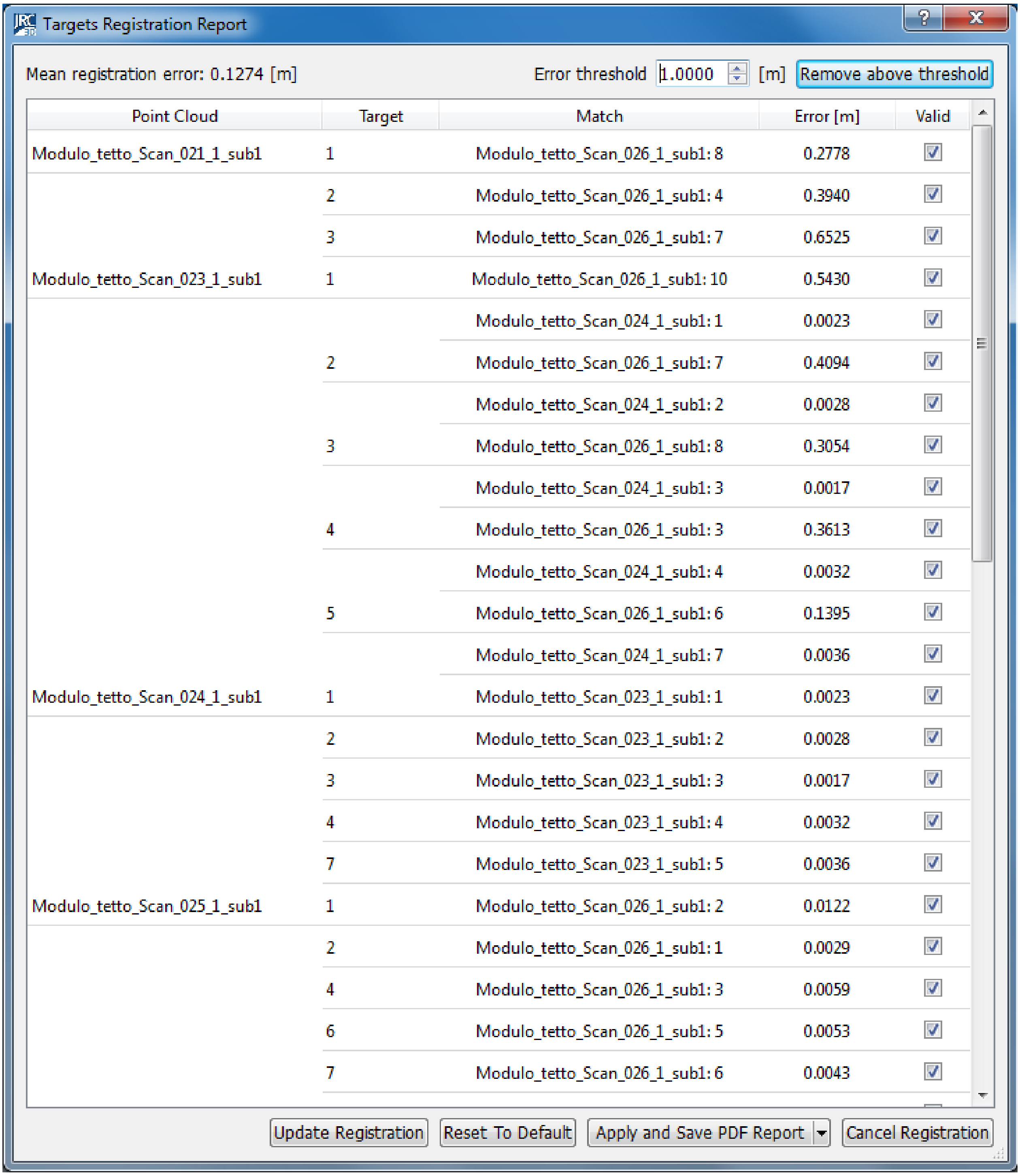
It is also possible to detect the spherical targets as a step in the Line Up Wizard. For example, you can import a set of tripod scans, pre-process them, detect the spherical targets, and run target-based preregistration on the clouds: the wizard will perform all these steps automatically for you. Just open the *Line Up Settings* and go to the *Preregistration* tab to adjust the desired settings for the steps of sphere detection and targets registration.

Regardless of the way you access the algorithm, after its execution Line Up's log window shows you how many spherical targets have been extracted, what is the average fitting error of the inliers to the found spheres, and what is the elapsed time. The average fitting error is a useful information to understand the degree of precision that you can expect by running target-based registration exploiting the found spheres.

Conditions and limitations

To be able to detect a sphere, the algorithm needs at least 30 points lying on the sphere. where each points has a scale of 13 mm. For example, if a 360° tripod scan has a width of 8000 points, at 18 m away from the origin there is a point every 13 mm. The closer to the origin the sphere is, the more points needed to detect it. However, you can still detect spheres with fewer points than required by using the *Find spherical target at a given point* button, described in the former section.

Targets Registration Report



Point Cloud	Target	Match	Error [m]	Valid
Modulo_tetto_Scan_021_1_sub1	1	Modulo_tetto_Scan_026_1_sub1: 8	0.2778	☑
	2	Modulo_tetto_Scan_026_1_sub1: 4	0.3940	☑
	3	Modulo_tetto_Scan_026_1_sub1: 7	0.6525	☑
Modulo_tetto_Scan_023_1_sub1	1	Modulo_tetto_Scan_026_1_sub1: 10	0.5430	☑
		Modulo_tetto_Scan_024_1_sub1: 1	0.0023	☑
	2	Modulo_tetto_Scan_026_1_sub1: 7	0.4094	☑
		Modulo_tetto_Scan_024_1_sub1: 2	0.0028	☑
	3	Modulo_tetto_Scan_026_1_sub1: 8	0.3054	☑
		Modulo_tetto_Scan_024_1_sub1: 3	0.0017	☑
	4	Modulo_tetto_Scan_026_1_sub1: 3	0.3613	☑
		Modulo_tetto_Scan_024_1_sub1: 4	0.0032	☑
	5	Modulo_tetto_Scan_026_1_sub1: 6	0.1395	☑
		Modulo_tetto_Scan_024_1_sub1: 7	0.0036	☑
Modulo_tetto_Scan_024_1_sub1	1	Modulo_tetto_Scan_023_1_sub1: 1	0.0023	☑
	2	Modulo_tetto_Scan_023_1_sub1: 2	0.0028	☑
	3	Modulo_tetto_Scan_023_1_sub1: 3	0.0017	☑
	4	Modulo_tetto_Scan_023_1_sub1: 4	0.0032	☑
	7	Modulo_tetto_Scan_023_1_sub1: 5	0.0036	☑
Modulo_tetto_Scan_025_1_sub1	1	Modulo_tetto_Scan_026_1_sub1: 2	0.0122	☑
	2	Modulo_tetto_Scan_026_1_sub1: 1	0.0029	☑
	4	Modulo_tetto_Scan_026_1_sub1: 3	0.0059	☑
	6	Modulo_tetto_Scan_026_1_sub1: 5	0.0053	☑
	7	Modulo_tetto_Scan_026_1_sub1: 6	0.0043	☑

This dialog allows you to refine and exploit the result of a target-based registration.
On top of the dialog, the *mean registration error* is shown, along with the editable *error threshold.*

In the central table of the dialog, the matched targets are listed, grouped by scan. Note that each target can have one or more match with targets from other point clouds.

You can unflag a matching target pair, by pressing the checkbox on the right, and press *Update registration* on the bottom left. The registration will be recomputed without the outlier and the mean registration error will improve accordingly.
Please note that the alignment between clouds might be lost when matches are unchecked. In any

case you can restore the original computed registration by clicking the *Reset To Default* button in the bottom part of the dialog.
Once you are satisfied with the computed alignment, you can apply the registration and choose to save a PDF report.

You can also unflag multiple matches at the same time by selecting an error threshold and clicking the *Remove Above Threshold* button in the top right corner of the dialog.

Georeferencing

Georeferencing means that the internal coordinate system of a cloud of points or a group of them or in general a group of item in the Reconstructor project can be related to a ground system of geographic coordinates or a local User Coordinate System.

It is recommended to use at least 4 points to georeference a model (or an entire project), so as to have redundancy and to avoid coplanarity on three points that could lead to an "upside down" georeferencing in particular conditions.

This icon menu groups the function written below

Point Cloud Georeferencing

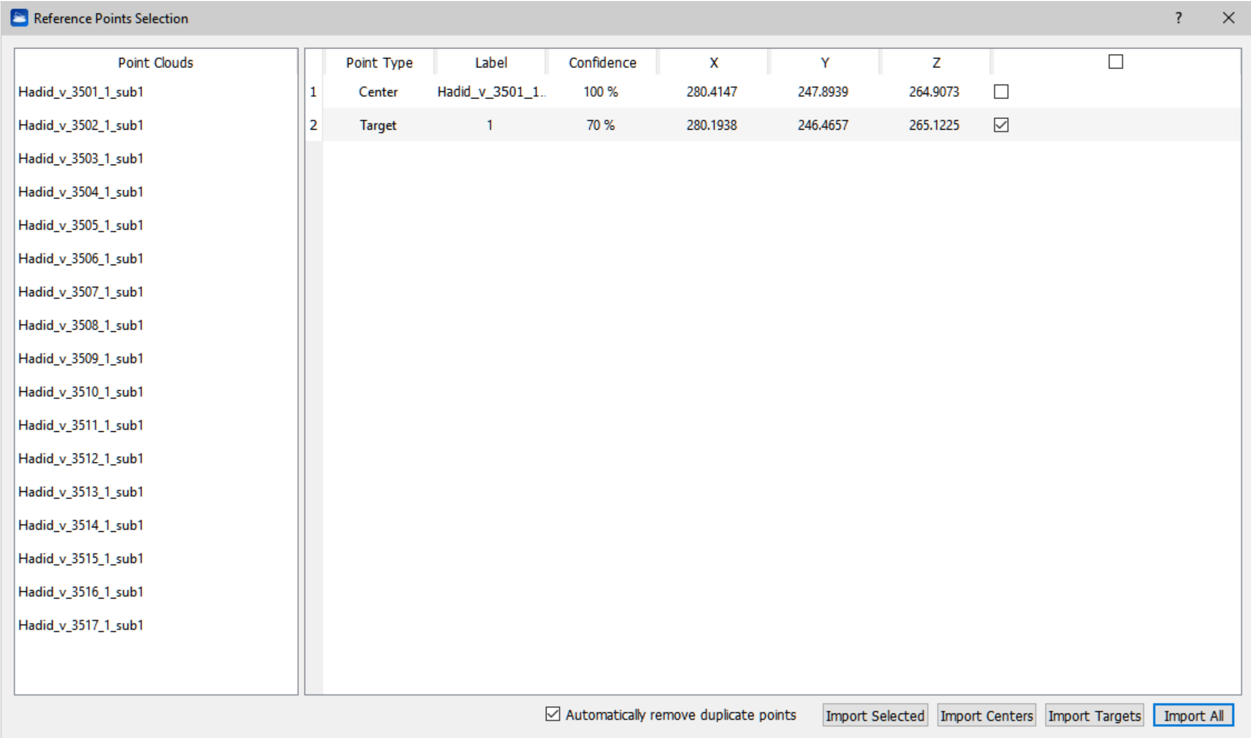
Point List Registration

Point clouds georeferencing

This function allows the *geo-referentiation* of multiple scans. The workflow is divided in two steps:

- Reference points selection
- Scans geo-referencing

Reference points selection dialog

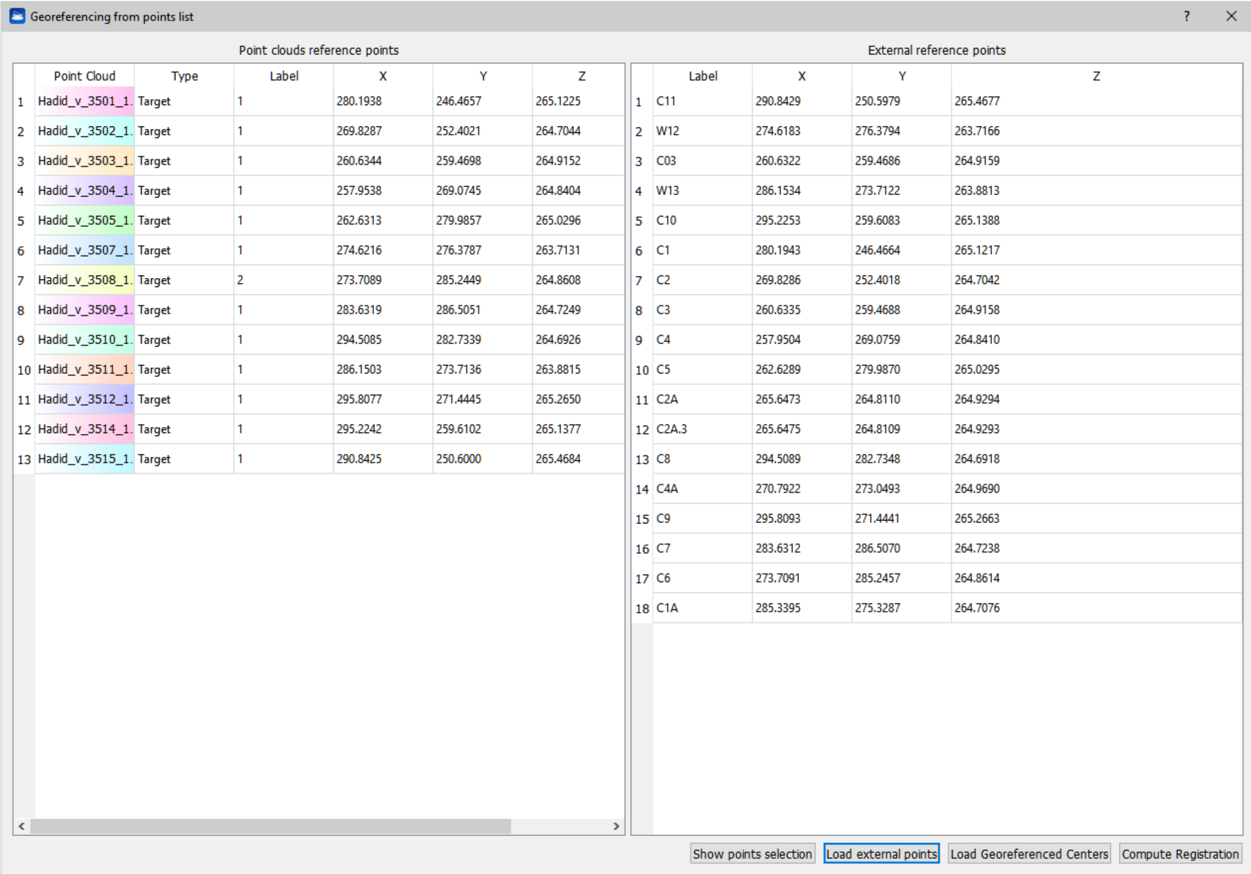


This dialog allows the selection of multiple reference points from the point clouds target and center points.
Target points can be automatically or manually set using the targets registration dialog or targets editor dialog tools.

The buttons in the bottom part of the dialog have the following functions:

- **Automatically remove duplicate points:** if two or more selected points are within a 0.2 meters distance between each other, only the higher confidence point is selected. This option can be used to pick only one target point if the same target was detected in more than one point cloud;
- **Import selected:** use only checked points for the geo-referencing step;
- **Import centers:** use only the point clouds center points for the geo-referencing step;
- **Import targets:** use only the point clouds target points for the geo-referencing step;
- **Import points:** use all the listed points for the geo-referencing step.

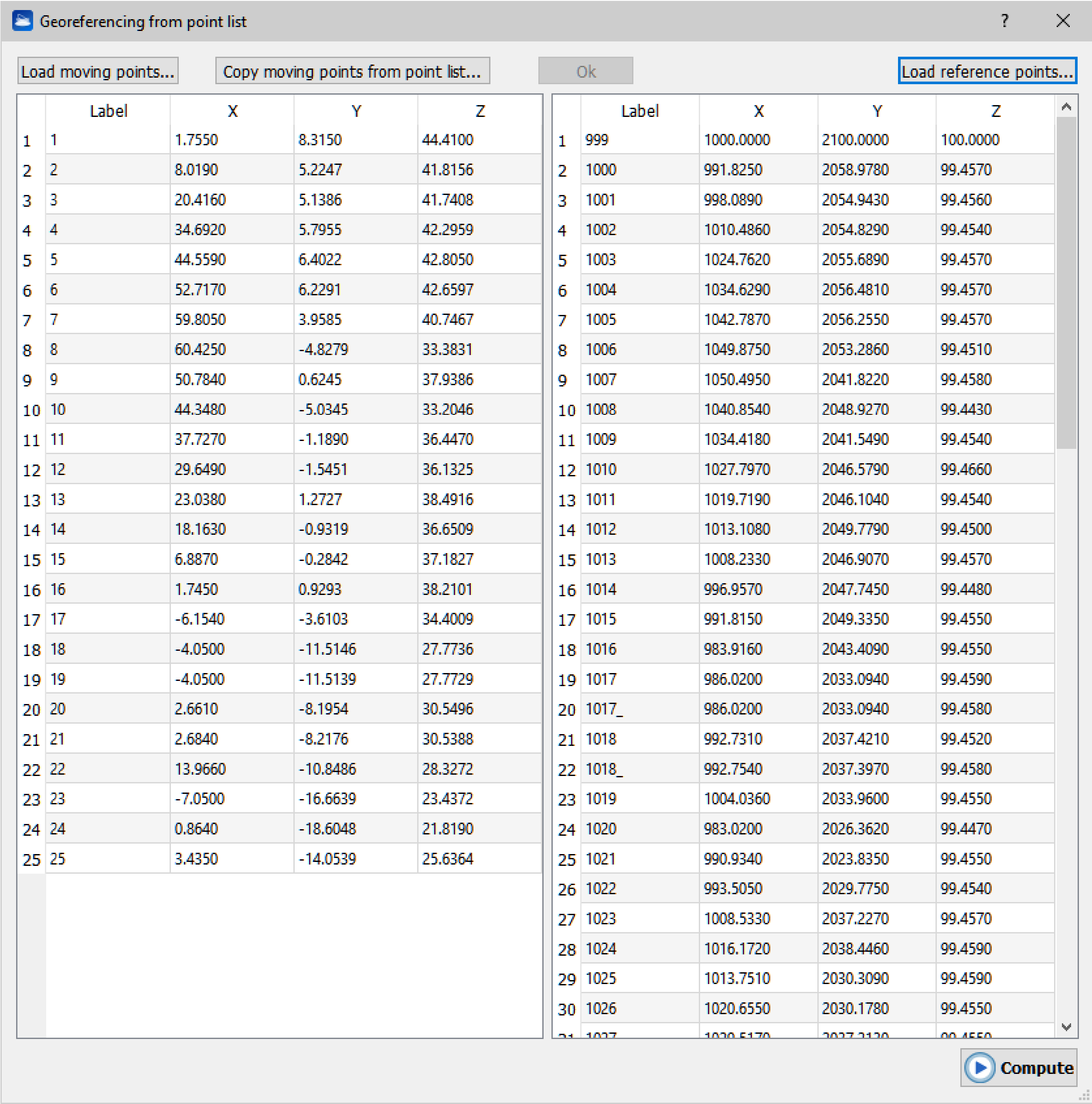
Scans georeferencing dialog



This dialog allows the user to load external reference points for geo-referencing scans using previous selected target points.
The buttons in the bottom part of the dialog have the following functions:

- **Show points selection:** opens the reference points selection dialog;
- **Load external points:** imports external point from a textual point list;
- **Load georeferenced centers:** imports the georeferenced center point of the point clouds if present. Georeferenced coordinates for the center point can be set automatically (depending on the format) or manually when importing the given point cloud;
- **Compute registration:** automatically finds the best matching points from the right and left lists and computes the alignment between them. See details in dedicated Compute Registration page.

Georeferencing from point list



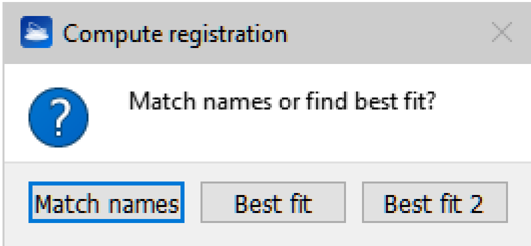
This dialog allows you to register a list of points, taken from a text file or from the point list window, against another list of points that you consider as reference. In particular, these last points can be *geo-referenced.*

The left half of the dialog regards the *moving* points. You can load them from a text file with the button *Load moving points*; in this case you are invited to select a text file and then the parse point list dialog appears. Otherwise, you can copy the points listed in the point list window via the button *Copy moving points from point list window*. Then, the moving points appear listed in the left table.
The right half of the dialog regards almost symmetrically the reference points. The button *Load reference points* on top right is needed to load the points from a text file, the right table hosts these points. To start the registration, the left table has to contain as many points as the right table.

When you have loaded the points, press *Compute* on the bottom right; the Compute Registration dialog will appear.

Compute Registration

After pressing the "Compute" button in Georeferencing procedure the following window will appear with three choices:



You can register points, scans, any items or a complete project by

- **Match names**: coupling points / targets according to their **labels**, no combination search is performed, the points / targets are coupled by name.
- **Best fit**: tries out all the possible pair combinations to find the best fit. Recommended for a number **less than 10 pairs** of points / targets.
- **Best fit 2**: tries out some smart combinations to find the best fit. Recommended for a number **greater than 10 pairs** of points / targets.

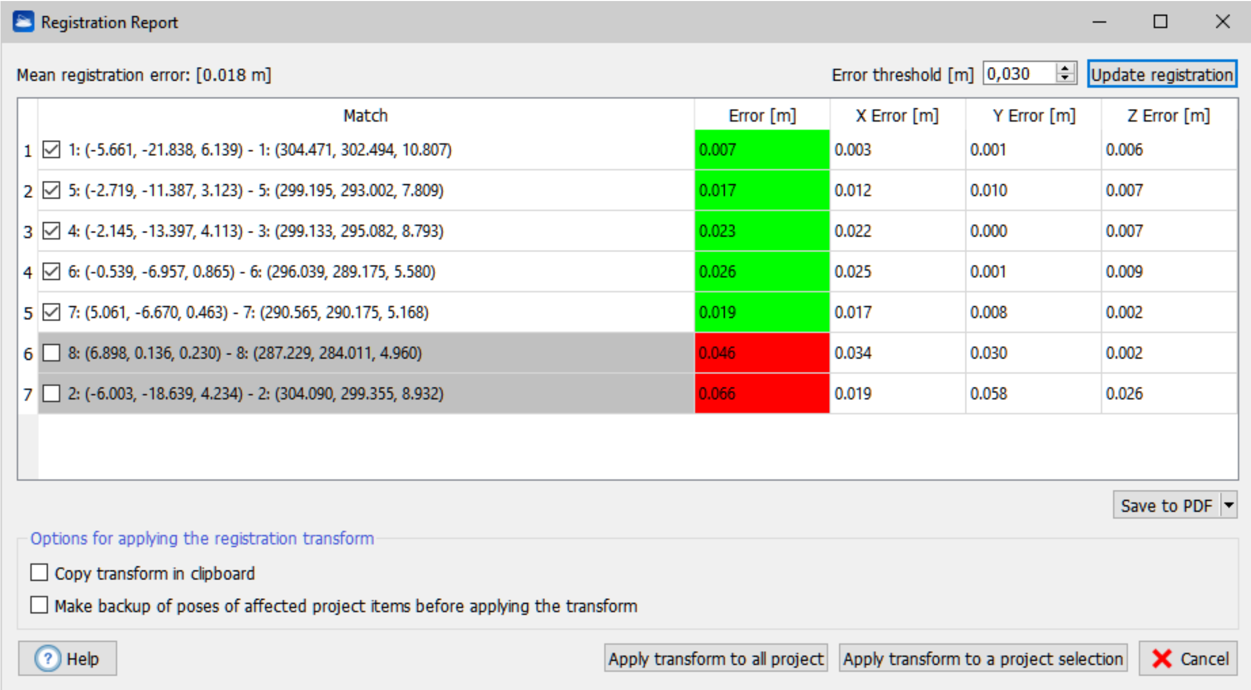
The Best fit adjustment procedure performs an optimization by minimizing the least squares errors to find the best rigid transformation between the moving and reference point set.
The best set of correspondences between those points is found by evaluating the rigid transformations between the triangles formed by every combination of three moving points against similar triangles in reference point set.

The first option is much faster but it assumes that you are sure about how to match your points.
After you have selected either *Match names, Best fit* or *Best fit 2*, you can refine and make use of the results of the registration, in the *Registration report dialog*.

The Registration Report dialog is shown at the end of the process.

For a general overview of Reconstructor®'s registration tools, see Registration techniques.
Return to the Georeferencing chapter.

Registration report dialog



This dialog allows you to refine and exploit the result of a registration. This dialog is used as final step of the following procedures: single-scan georeferencing, georeferencing from point list, and platform calibration.

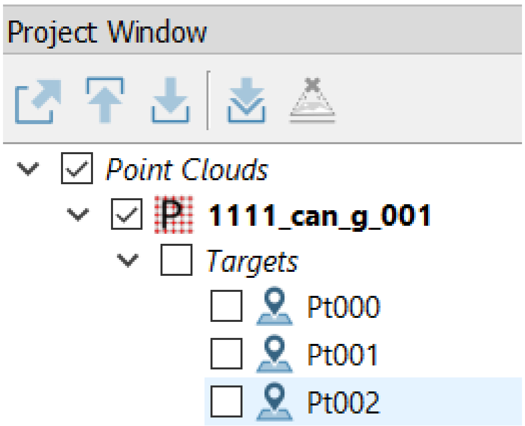
- On top of the dialog, the *mean registration error* is shown, along with the editable *error threshold.* Also errors according the current UCS components are shown.
- In the central table of the dialog, the point pairs are shown along with their associated registration errors. Errors above the error threshold are displayed in red, and you could consider the corresponding pair as outlier. If you uncheck an outlier pair, by pressing the checkbox on the left, and press *Update registration* on top right, the registration will be recomputed without the outlier and the mean registration error will improve.
- In the bottom part of the dialog, there are options on how to make use of the obtained registration. The option *Copy transform in clipboard* is useful for having a backup of the registration transform to use on project entities that will be created at a successive time. The option *Make backup of poses of affected project items before applying the transform* will save the poses of the affected project entities in the *Exports* folder of the project, these poses can be restored later via the matrix transform tool, to undo the effects of the current registration. To access this tool, go to the project window, select the entities you want to register, right-click and choose *Registration -> Matrix transform* from the context menu.
- The button *Apply transform to all project* on the bottom extends the effect of the registration to all the project entities. All your project items (including annotations) will be moved according to the roto-translation defined by the registration. The button *Apply transform to a project selection* will pop up a dialog to select a subset of the project items. Only those selected items will be moved.

Targets, annotations and points conversion

It is possible to manage Targets, Annotations and Points inside the project, in order to change their role.
The conversion tools are also available in the context menu of the items.

- Target -> Annotation
- Annotation -> Target
- Target -> Point
- Point -> Target
- Annotation -> Point
- Point -> Annotation

Here an example of application: you picked a point in the 3D scene and you want to use it as target coupled with a defined 3D model. Right-click on the Point in the Project Window and select *Point -> Target*; a dialog will ask you which cloud you want to match the target. At the end of the process the Target will be



Remember that a Target is not an isolated entity, it is always combined with a point cloud (grid or unstructured) or a mesh as its sub-item.

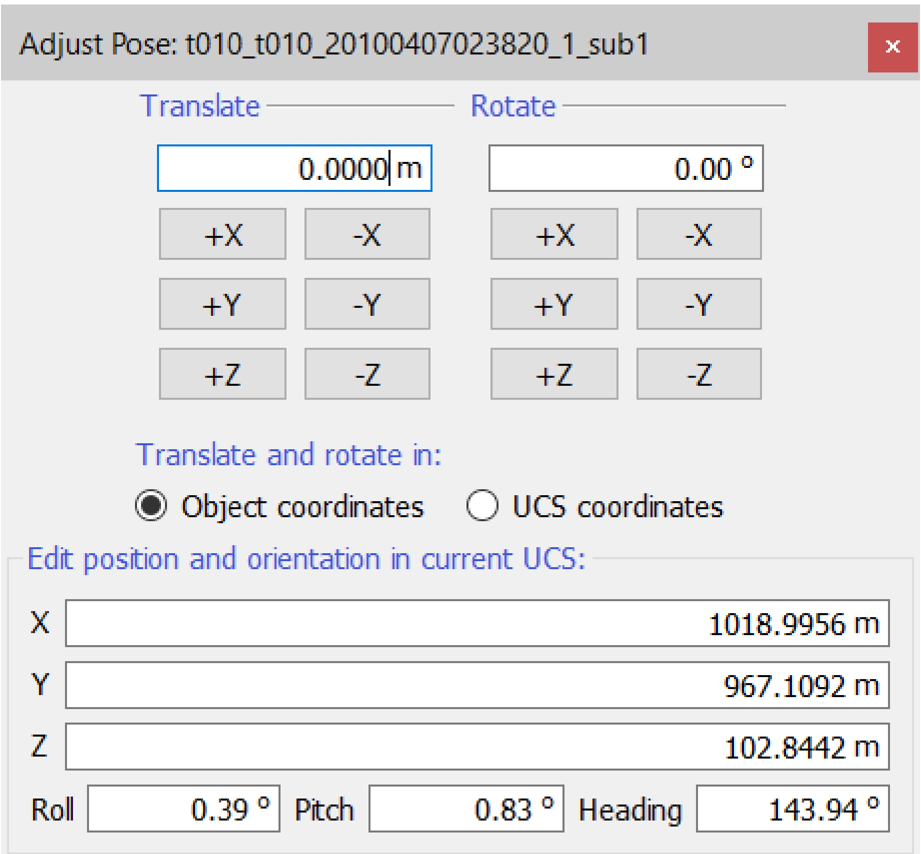
UCS & positioning

Reconstructor® includes various posing techniques:

- Manual Positioning
- Pose Transform
- Pose Dialog (contextual menu)
- Determine the cloud's vertical direction
- Create/edit UCS
- Create UCS from point list
- Create UCS from this pose (contextual menu)
- Define Project Reference System

Manual positioning (Adjust Pose)

The command is available in the *UCS & positioning* Top Toolbar tool, in the item's context menu and by pressing [Ctrl+A] keys.



It is used to manually change the pose of an object.

This window is enabled only if an item in the Project Window is selected. Then also the bounding box of the current item is highlighted in bold yellow.

Insert the desired quantity of translation (meters) and rotation (degrees) and press the positive and negative buttons +/- X, +/- Y, +/- Z to apply transformation.

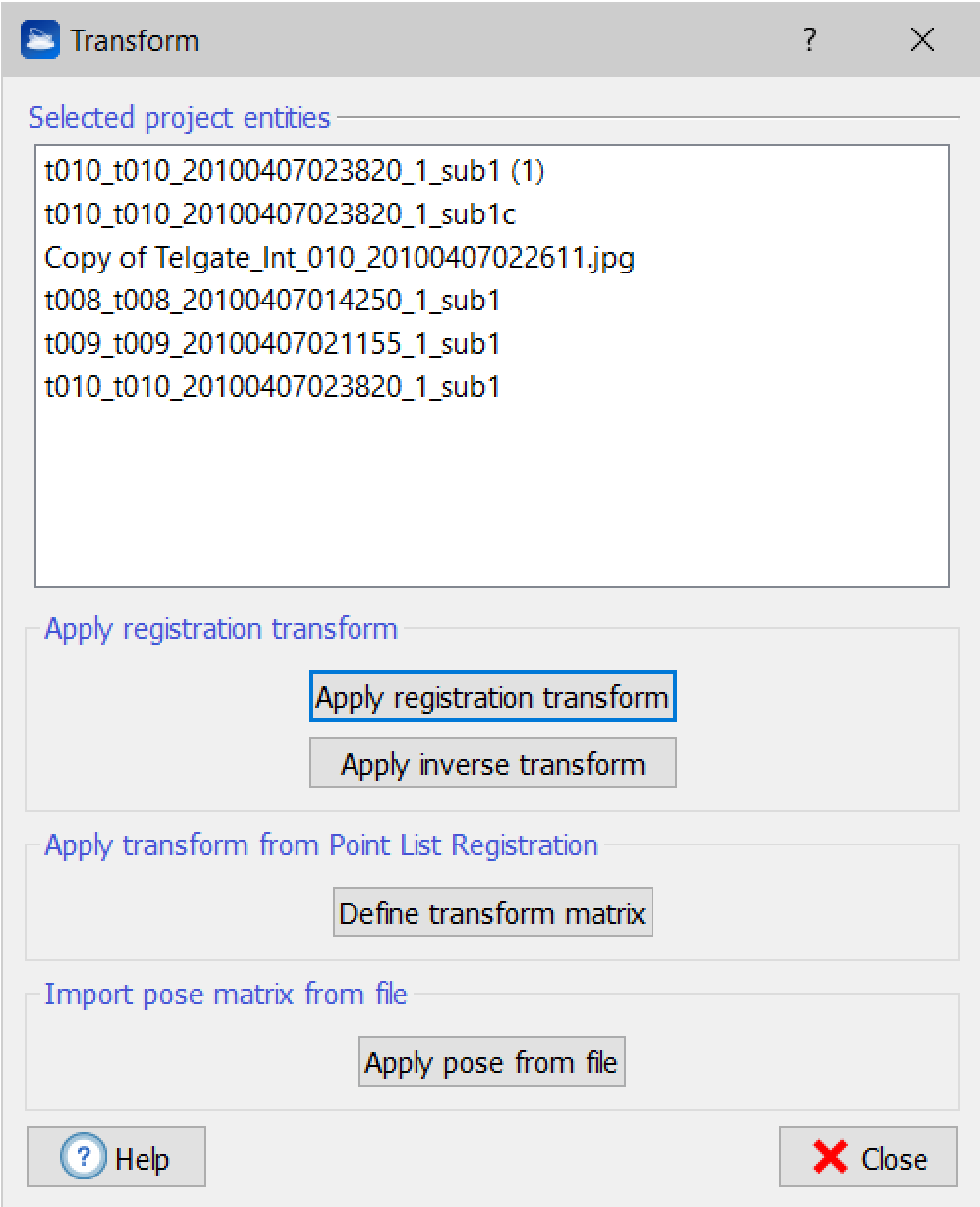
If **Object coordinates** is enabled, the rotation is around the origin of the object's coordinate system and the translation is along the axes of the same system.

If **UCS coordinates** is enabled, the rotation is around the current UCS' origin and the translation is along the axes of the same UCS.

See also the Pose dialog.

Pose Transform

The command is available in the *UCS & positioning* Top Toolbar tool and in the item's context menu.



The *Pose transform* command allows you to apply to your project entities the transforms coming from registration functions.

In pre-registration, registration, and geo-referentiation, you can copy in the clipboard the transform that results from the computation.

- The *Apply registration transform* button searches the clipboard for such a transform, if found, the transform is applied to multiple models.
- The *Apply inverse transform* button undoes the effect of the first button. It reads from the clipboard a registration transform and applies its inverse to the selected project entities. It's useful if you don't like the results of a registration.
- The *Define transform matrix* button allows you to manually define a transform matrix, that is post-multiplied to your project entities. The transform must be defined as referring to

the current UCS. A common situation in which this function is needed is when you have done a point list registration and copied the resulting transform to the system clipboard. In this case, to apply the transform press *Define transform matrix*, then *Paste matrix* and *Commit.* The selected project entities will be moved according to the registration transform matrix in the clipboard.

- The *Apply pose from file* button allows you to import the *.pose* files you export with E*xport pose*. “Export pose” is a command reachable from the context menu of any project item. It is designed to save the positions of your project items for backup. With *Apply pose from file*, you can therefore move your models to positions you saved in backup files.

Note: The pose matrix exported by *export pose* is different from the pose matrix displayed in the Pose dialog. The "export pose" function is designed to provide to the user the possibility of making a backup of the items' positions and to restore them at a later time in case something goes wrong. Now, if "export pose" would save the pose referred to the current UCS (that you see in the Pose dialog), what would happen if the user by mistake moves or deletes that UCS? The possibility of restoring the items' positions would be lost. Therefore, "export pose" exports another pose referred to an internal, hidden, never-changing Reconstructor global reference system.

Difference between Apply registration transform and Define transform matrix

This dialog has two buttons, *Apply registration transform* and *Define transform matrix*, that apparently do the same thing: getting the data stored in the system clipboard, interpreting them as a “registration transform” (a description of how move around models) and applying the transform.

The difference is as follows:

- *Apply registration transform* is suited for transforms calculated between Reconstructor®'s project entities,
- *Define transform matrix* is suited for transforms coming from other tools, or transforms calculated between entities not belonging to Reconstructor®'s project.

That is why you can use *Apply registration transform* only after pre-registration, registration, and geo-referentiation. These three tools, in fact, calculate a registration defined on Reconstructor®'s project entities, and therefore the registration transform is defined in a very precise and error-resilient way that is internal to Reconstructor. For example, if you do a registration, change UCS, and use A*pply registration transform* you will still get the correct result with all the decimal precision of the registration algorithm, independently of the UCS. On the other hand, after a point list registration you cannot use *Apply registration transform* and you have to use *Define transform matrix.* This is because *point list registration* works with *any* lists of points, in principle also not coming from Reconstructor®'s models, but from other sources like a total station or the Internet. In this case, having lost the relation with the current project, Reconstructor® cannot generate a precise and error-resilient registration transform of the first type and therefore *point list registration* returns a generic 4x4 transform matrix whose effect changes if the current UCS changes.

In conclusion, there are two buttons because there are two types of registration transforms: a more specific and more precise one, and a more general-purpose one.

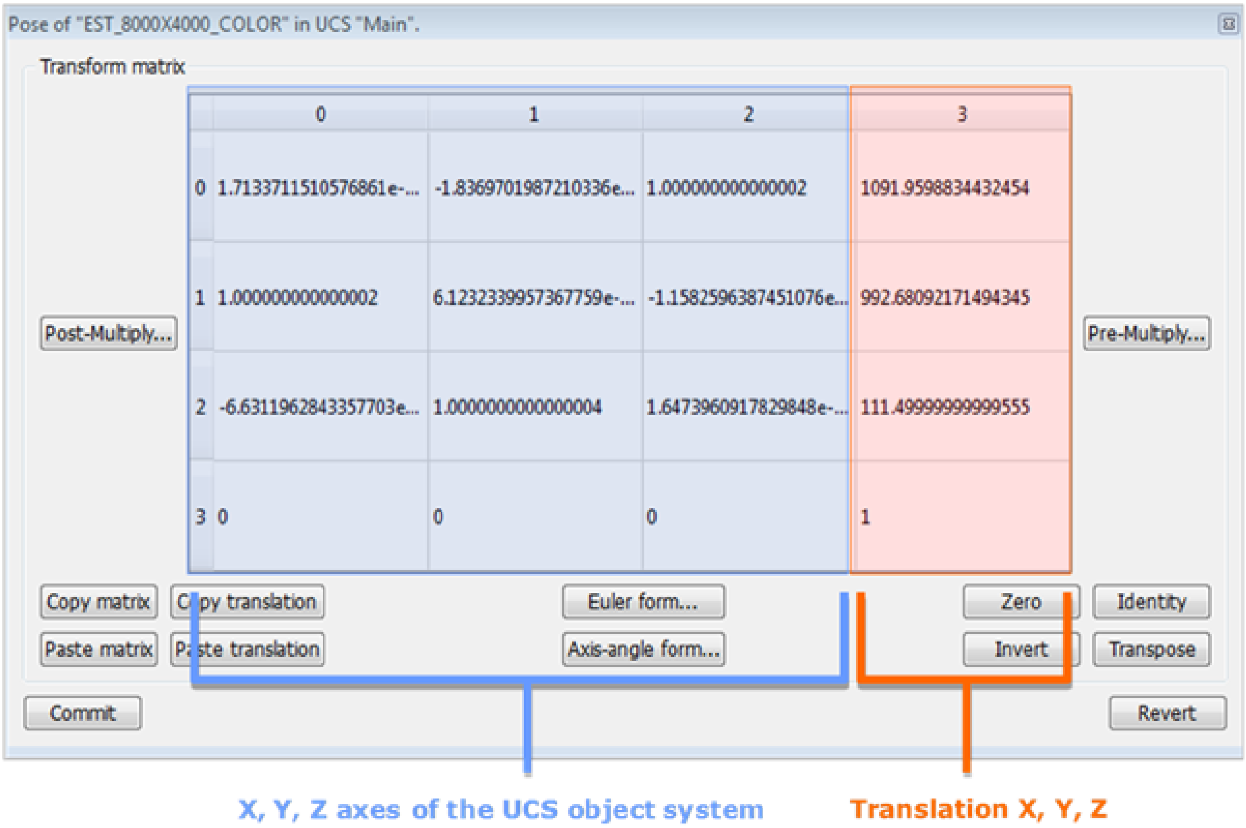
See also Pose dialog.

Pose

The command is available in the *Data Management* > *LineUp* Top menu and in the item's context menu *Pose&Registration*.

In Reconstructor®, the 3D position of each object (except annotations) is described by the **pose** matrix (*transformation matrix*).

You can manage different UCS systems (local or global/geo-referenced).
A *transformation matrix* defines how a project entity is located and oriented with respect to the current *UCS*. Therefore, having knowledge about transformation matrices, it is possible to perform specific and precise translations and rotations on a project entity.

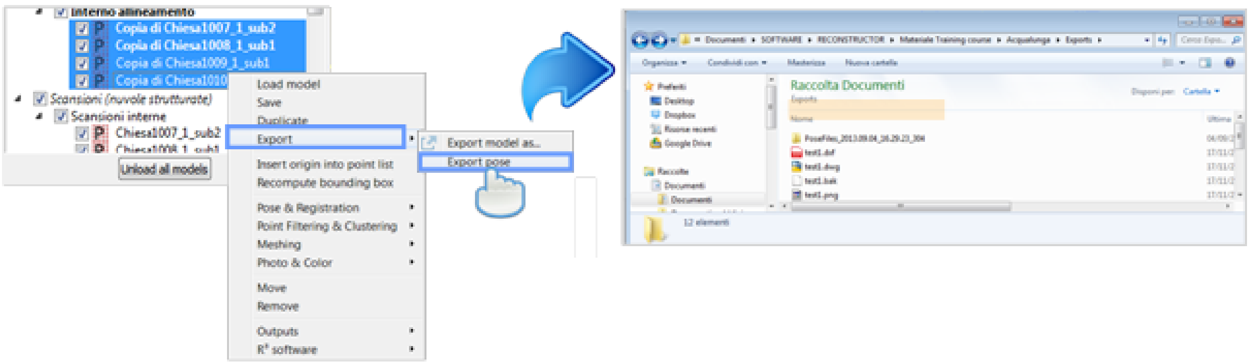


A *transformation matrix* is a 4x4 matrix that specifies how an object *O* is located and oriented with respect to a global coordinate system. The first, second and third column of the transformation matrix contain respectively the versors of the *X*, *Y*, and *Z* axes of the system anchored with the object *O*. The fourth colum contains the translation vector of *O*'s origin with respect to the UCS' origin. The fourth row of the translation matrix is always (0 0 0 1).

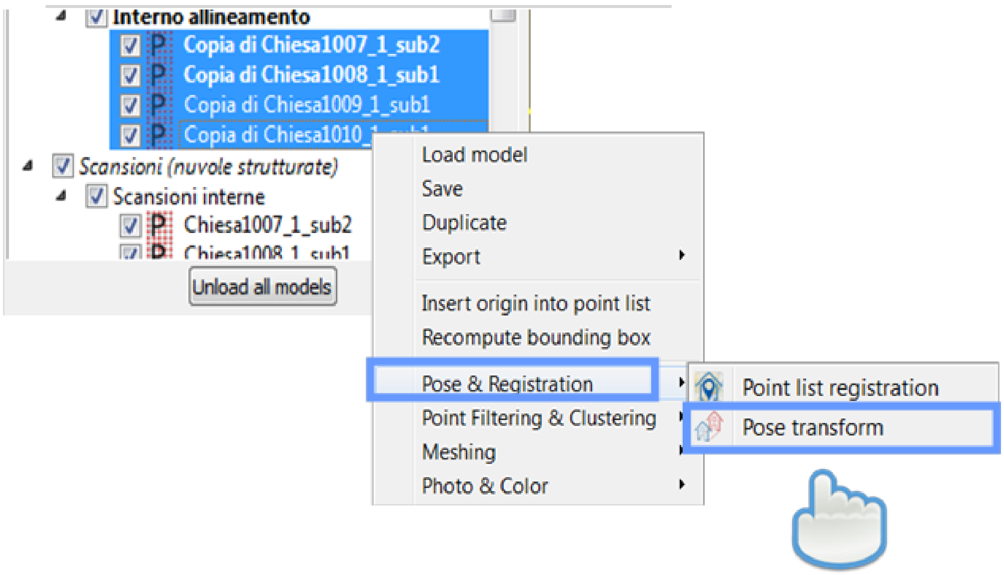
See also Manual positioning (Adjust Pose).

Restore a Pose

You can restore an object position in case of registration and geo-referencing error once the Pose file is saved.
To save the Pose you have to export it:

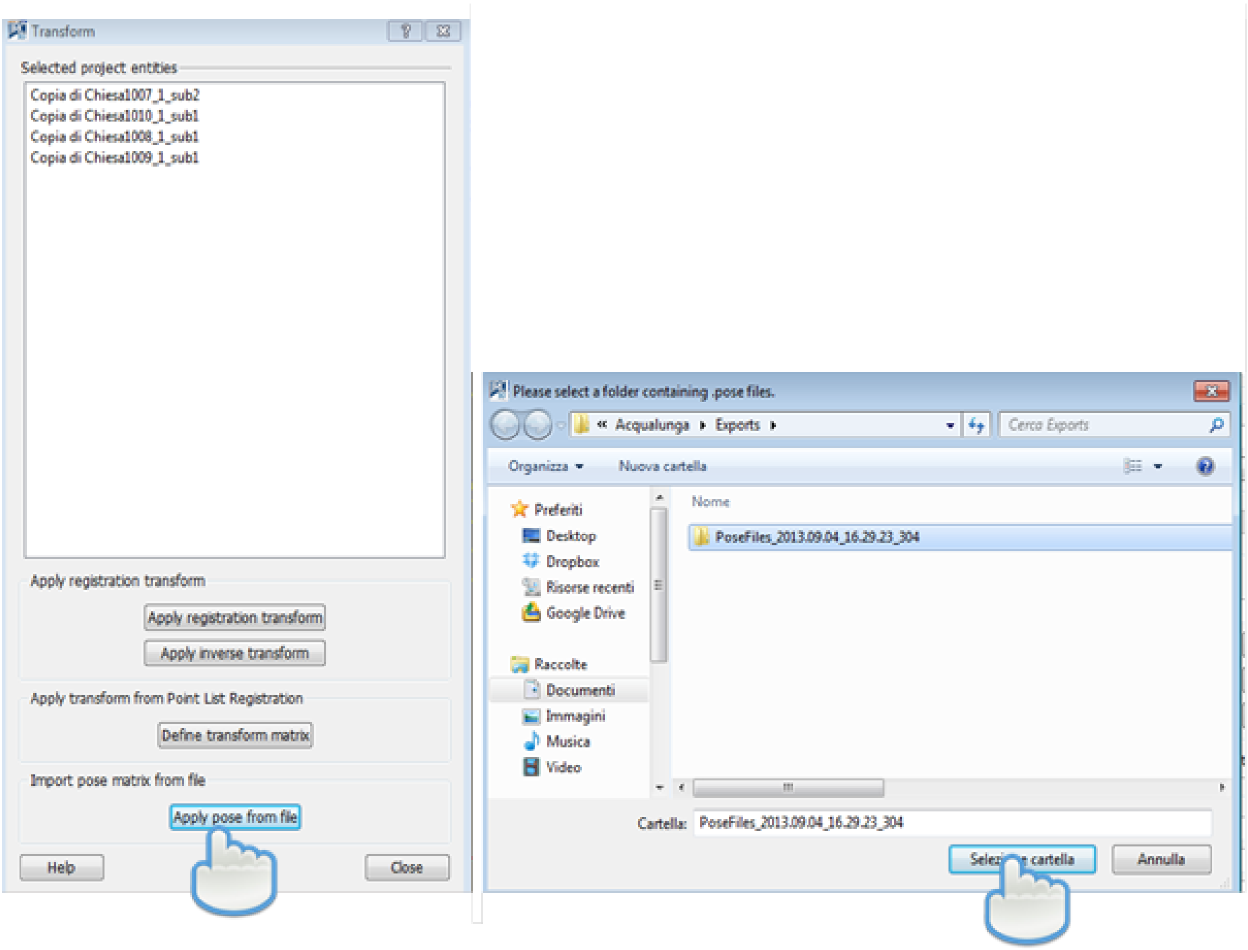


The software automatically creates a folder *PoseFiles_2013.09.04_16.29.23_304* that contains the Pose exported. This folder is saved in the Exports folder and the name contains the date (day and time) of the creation.
Apply the Pose file by selecting the objects desired → right mouse button → Pose&Registration → Pose transform.

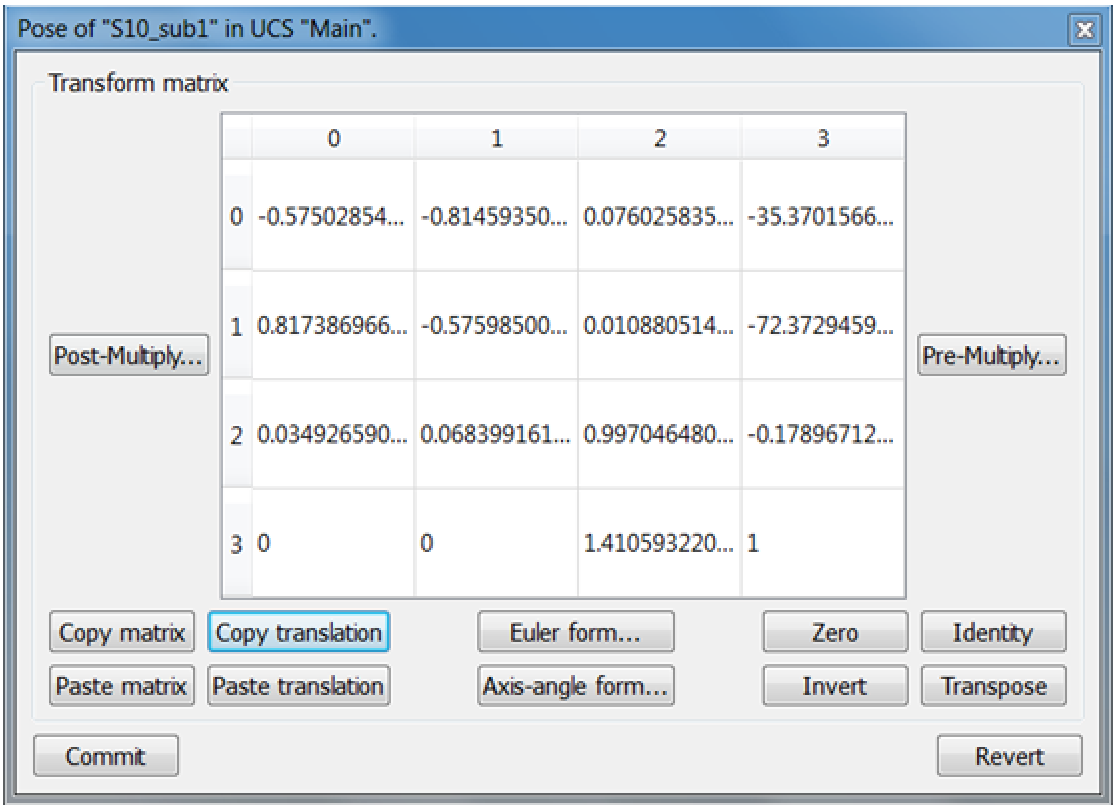


A window appears. Select Apply Pose from file and select the Pose you want to restore.

Items must have the same name of the pose file to apply pose.



Advanced Options



This dialog is designed for advanced users, since it requires specific knowledge of 3D computer graphics, in particular knowledge about *transformation matrices*.
In the dialog there is a central 4x4 table displaying the transformation matrix of the current project entity. On the sides, the buttons **Post-multiply** and **Pre-multiply** are located.
If M is the current matrix, a pre-multiplication by matrix P is defined as

$$M' = M \cdot P$$

If M is the current matrix, a post-multiplication by matrix P is defined as

$$M' = P \cdot M$$

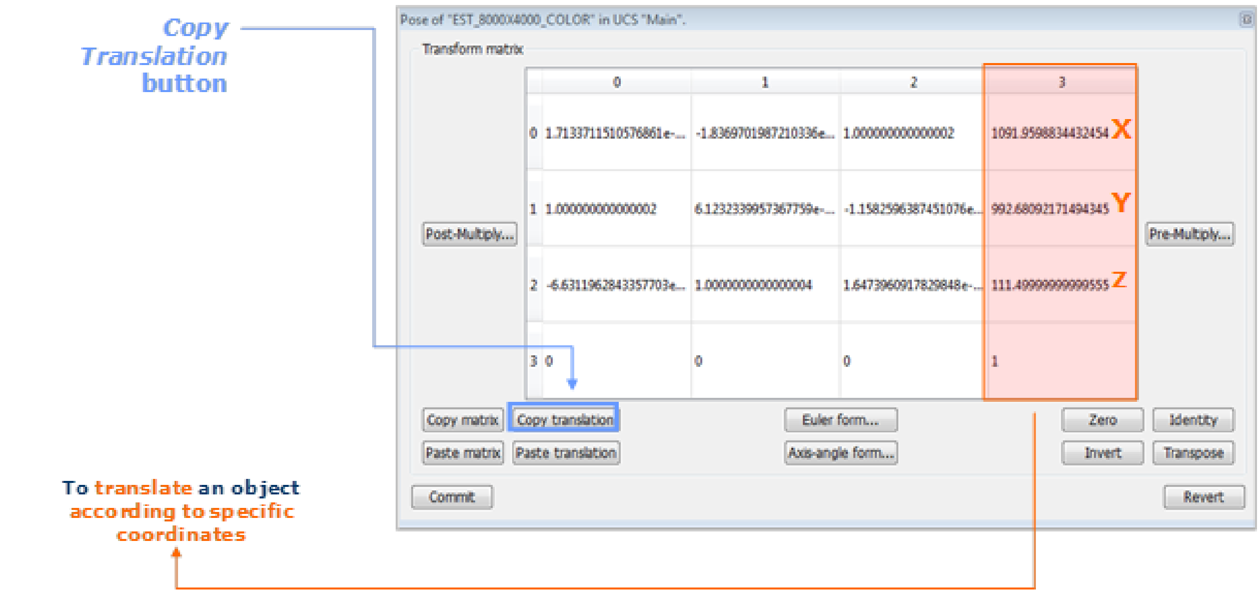
Any number of pre-multiplications or post-multiplications to the current matrix is possible.
Confirm any modification of the matrix by pressing **Commit** on bottom left of the dialog. Press **Revert** to discard any modification. The upper-right Close button (x) has the same effect as Revert and hides the dialog.
To speed up the definition of the matrix the following operations are available:

- **Copy in clipboard**: copies the current matrix in the clipboard, useful for export or to paste it to another model
- **Paste from clipboard**: pastes the matrix from the clipboard, confirm by pressing Commit
- **Euler form**: define the transformation as rotation plus translation. The rotation is defined as a sequence of rotations along the local coordinate frame axis. The Pitch angle is around X, Heading is around Y, Roll is around Z, in degrees. If the sequence is YXZ, then first the rotation around Z(Roll) is applied, then X(Pitch), then Y(Heading). Press OK and the homogenous transformation matrix is computed.
- **Axis-angle form**: define the transformation as rotation plus translation. The rotation is around the desired axis (X,Y,Z) direction by amount Angle, with the right hand curl rule. Press OK and the homogenous transformation matrix is computed.
- **Zero**: clears the matrix to zero
- **Identity**: defines the matrix to be the identity
- **Invert**: inverts the current matrix
- **Transpose**: transposes the current matrix

For example

Rotation around X axis by 90 degrees	Rotation around Y axis by 90 degrees	Rotation around Z axis by 90 degrees
$\begin{vmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & -1 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{vmatrix}$	$\begin{vmatrix} 0 & 0 & 1 & 0 \\ 0 & 1 & 0 & 0 \\ -1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{vmatrix}$	$\begin{vmatrix} 0 & -1 & 0 & 0 \\ 1 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{vmatrix}$

Move objects with the Roto-translation Matrix

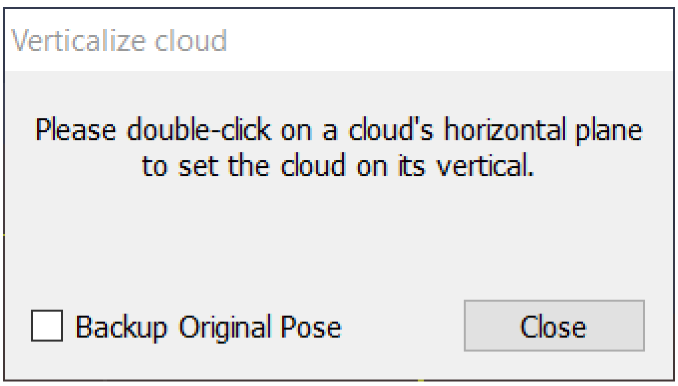


Copy Translation button: To copy the object translation (column 3).
Use this option to translate an object on these coordinates.
1. Press Copy Translation
2. Open the Pose of the object to move
3. Select Paste translation

To translate an object according to specific coordinates, type the desired coordinates in the column 3 (first line stands for X, second Y, third Z).
The translation will be made according to the object's UCS reference system.

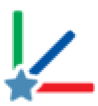
Determine the cloud's vertical direction

The command is available in the *UCS & positioning* Top Toolbar tool.

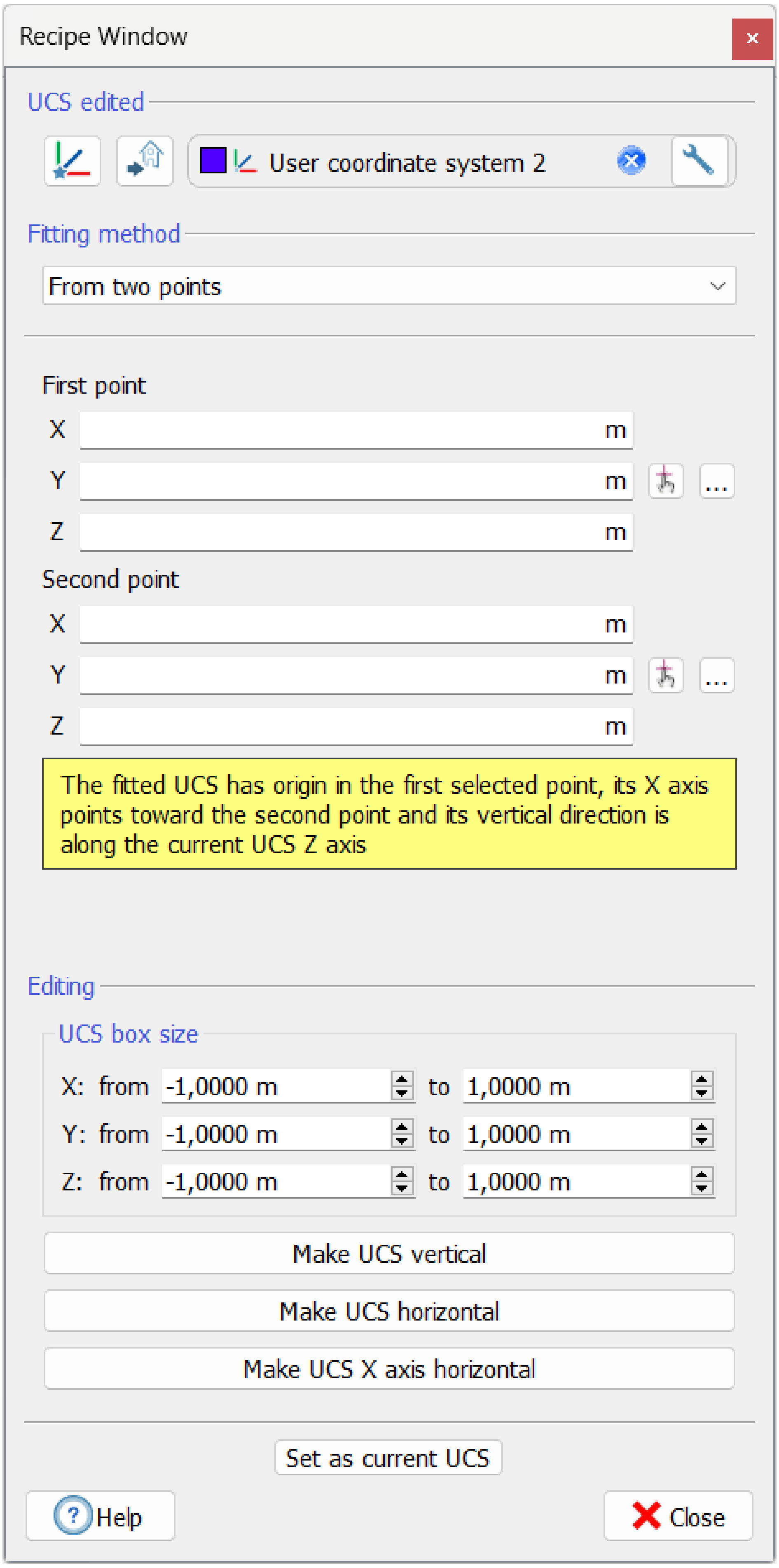


It defines the vertical direction of the point cloud by defining a plane that should be horizontal (e.g. a floor) - by double clicking on it - and setting the vertical direction of the point cloud (z axis) as the normal direction of this plane, rotating the point cloud.

Create/edit UCS



The command is available in the *UCS & positioning* Top Toolbar tool.



The *Create/edit UCS* command allows you to create a new UCS (by pressing) and manage UCS already present in the project (by dragging&dropping it in the top of the dialog).

button allows to modify the UCS's name.

Note that it is not possible to edit the *Main* UCS.

Fitting Methods

Six methods are available to create and modify an UCS.

From two points

The UCS is here defined by setting 2 points in the 3D space:

- UCS origin
- a point on the X axis

The fitted UCS has origin in the first selected point, its X axis points towards the second point and its vertical direction is along the current UCS Z axis.

These points can be selected directly by double clicking on the 3D scene, chosen between the Points listed in the project or identified by manually entering their coordinates.

From three points

The UCS is here defined by setting 3 points in the 3D space:

- UCS origin
- a point on the X (or Y axis)
- a point on XY plane

These points can be selected directly by double clicking on the 3D scene, chosen between the Points listed in the project or identified by manually entering their coordinates.
At the end of the process it is possible to invert the Z direction.

From Current view

The UCS is adapted to current view, with the origin in the point of view's origin, the Z axis oriented towards your eye and X, Y axes parallel to the 3D window edges.

Fit UCS to faces

The UCS is here created by selecting portions of the point clouds to define both the UCS's XY plane (horizontal) and the UCS's XZ plane (side).
To do it the lasso command is used to select the points on a "flat" [suggested] surface.

This method is particularly useful when you have to create an UCS whose planes are parallel to the floor and walls, in order to extract orthophoto, cross sections, ...
Check that your points sample is large enough so you don't make any serious mistakes.

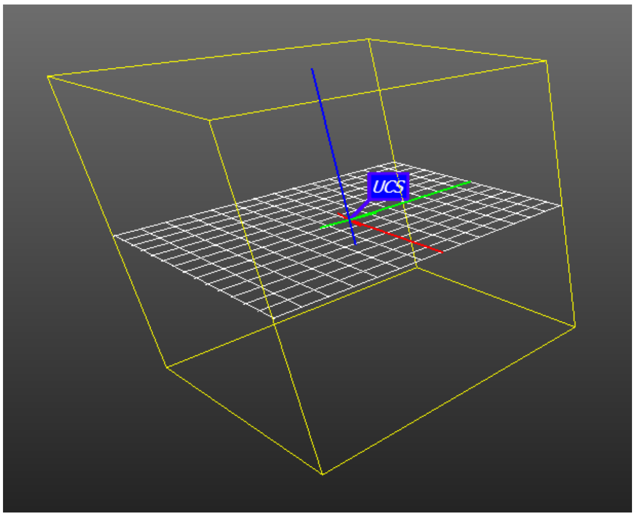
From an item's pose

The UCS of the desired item is cloned. Drag&Drop the item in the dialog or double click on it in the 3D scene.

From points registration

The UCS is created from two sets of points.
See Create UCS from point list.

Editing



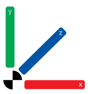
By default the UCS's bounding box dimensions are set between -1 m and +1m, in all the 3 main directions (X,Y, and Z), but you can easily set different values, in order to make more visible the UCS at large distances, for example.

By clicking the buttons below, you can

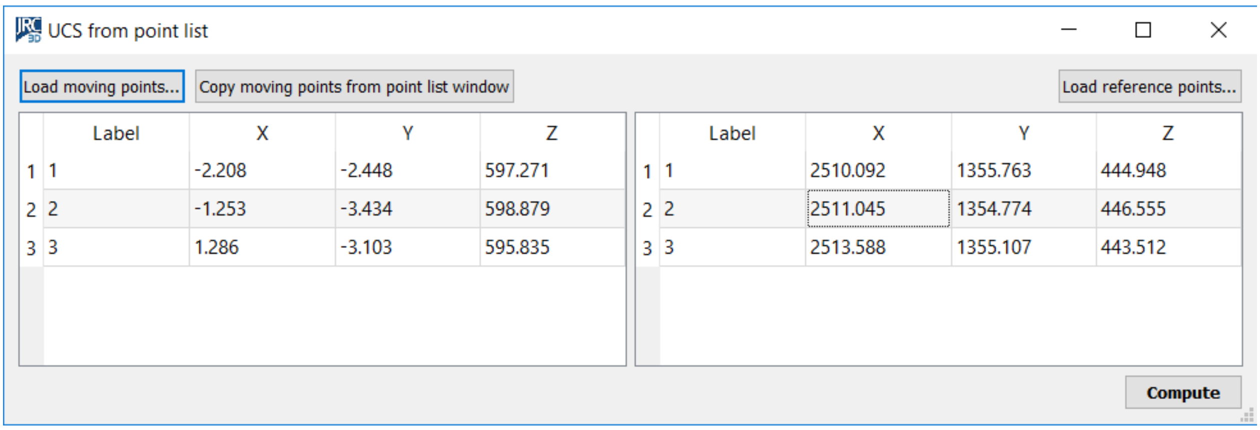
- *Make UCS vertical*, setting its Z axis parallel to the Z axis the current UCS
- *Make UCS horizontal,* setting its XY plane parallel to the XY plane of the current UCS
- *Make UCS X axis horizontal,* setting the horizontality of the only X axis.

At the end you can choose to set the just created/modified UCS as current in the project, by pressing the *Set as current UCS* button.

Create UCS from point lists



This command creates a new UCS from two sets of points. Switching to the resulting UCS will show the points of the first set with the coordinates of the second set.



	Label	X	Y	Z
1	1	-2.208	-2.448	597.271
2	2	-1.253	-3.434	598.879
3	3	1.286	-3.103	595.835

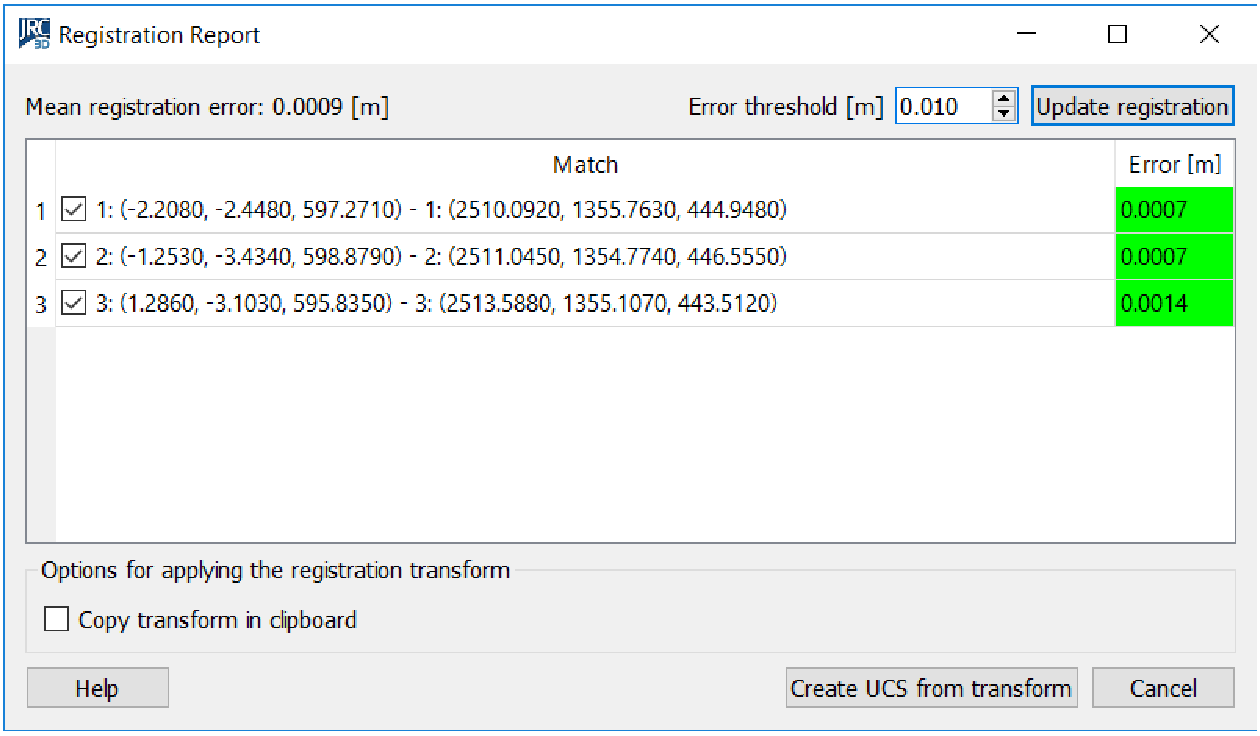
	Label	X	Y	Z
1	1	2510.092	1355.763	444.948
2	2	2511.045	1354.774	446.555
3	3	2513.588	1355.107	443.512

This function is accessible from the Navigation menu, or from the toolbar of the point list window.

Given two lists of points, this function creates a new UCS such that the points having coordinates in list 1 will assume the coordinates in list 2 after switching to the resulting UCS, except from a controllable mean-squared error.

Press the buttons *Load moving points* on top left to load a text file with the first point list, and *Load reference points* on top right to load a text file with the “reference” coordinates of the points, the coordinates that are going to be shown in the new UCS. To parse the text files with the point list, an appropriate tool is used. You can use as moving points the points listed in the point list window, via the appropriate button on top left.

After loading the first and the second list of points, press *Compute* on the bottom right. A dialog appears, asking whether you want to register the points by coupling them according to their labels, or by trying out all the possible pair combinations to find the best. The first option is much faster but it assumes that you are sure about how to match your points. After you have selected either *Match names* or *Best fit*, you can refine and make use of the results of the registration, in the dialog shown below.



In the dialog above, it is possible to set the error threshold and to exclude outliers in order to improve the mean registration error. The way to do that is described here (ICP Registration Dialog). Once you have refined the mean-squared error (shown on top left), press *Create UCS from transform* to create the desired UCS. The resulting UCS will be inserted in the project window.

This procedure uses the same calculations as georeferencing from point list.

Create UCS from point list registration: starts a procedure to create a coordinate system that is based on the algorithm of the former procedure. It exploits the same concept of registering the listed points against another point list that is considered as reference. However, the final result does not consist in moving all the project items, but in creating a new UCS and to place it so that all the other entities appear as if they were moved according to the found registration. For example, if the registration implies a translation of (1000, 1000, 1000), then a new UCS is created placed (-1000, -1000, -1000) away from the current one. Then, this new UCS is set as current. One benefit of this procedure is that you can easily undo its effects, by simply setting the former UCS back as current. The drawback is that ambiguities may rise if you work with other users that do not use the same UCS and assume that you have geo-referenced the data, or ambiguities may rise if you import Reconstructor models saved in earlier projects that were geo-referenced in the other way.

Create UCS from this pose

The command is available in the item's context menu.

The context command *Create UCS from this pose* allows to create a new reference system – UCS according to any item in the project.

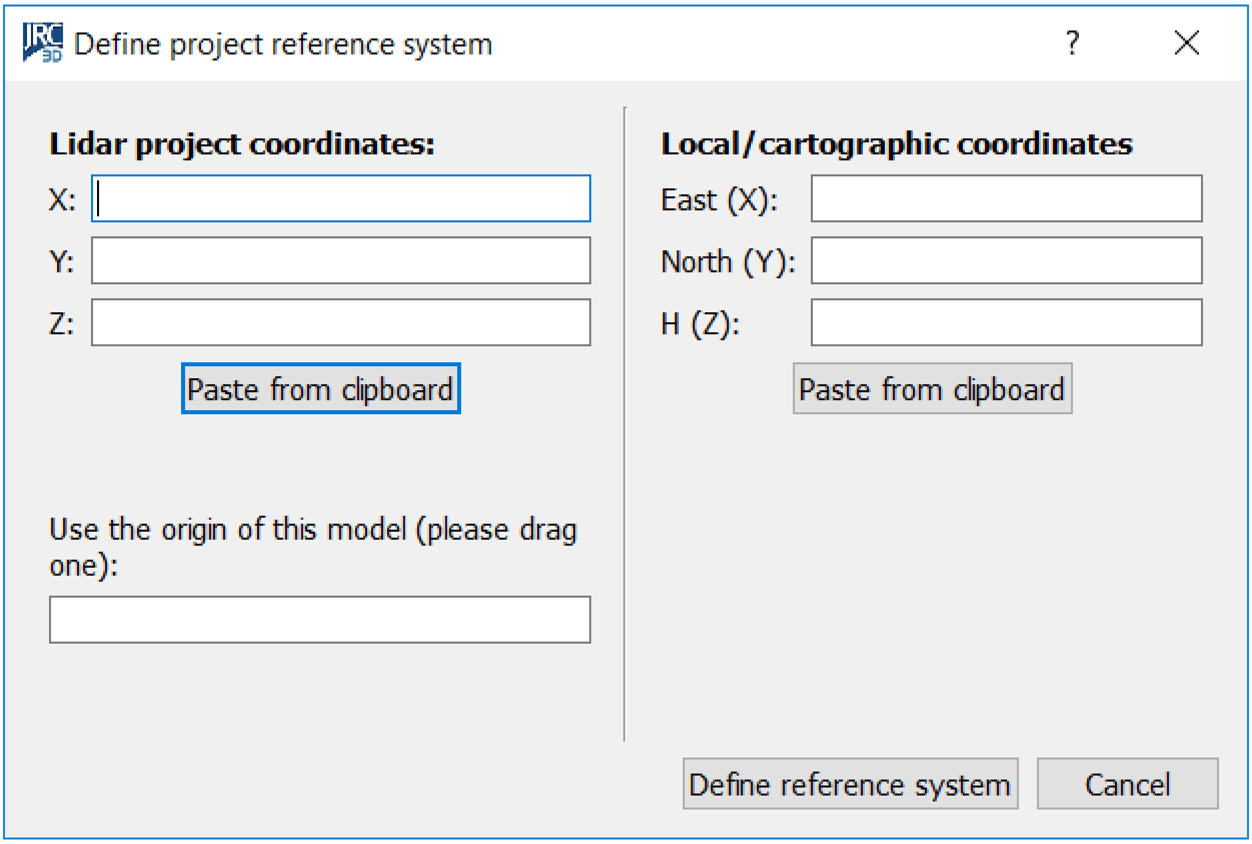
It could be useful to create an UCS according to an object in order to help a particular visualization.

Here an example.

Use a vertical plane fitting a wall to create a point of view (and so an orthocamera, e.g.) in front of the façade:

- define the plane
- in the plane's context menu click on *Create UCS from this pose*: a new UCS is created and stored in the folder *User Coordinate Systems*.
- to set the new UCS as *current*, select the new UCS and *Set as current* (the current UCS appears in bold).

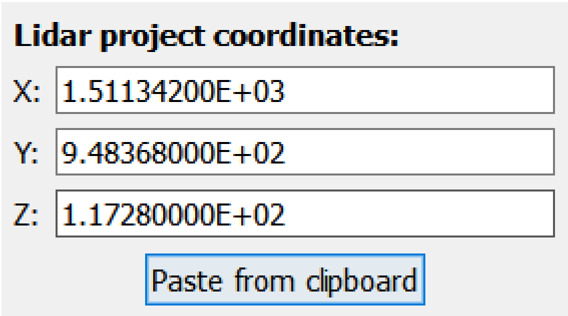
Define project reference system



This dialog allows you to quickly define a new UCS by just specifying one point's coordinates in the old and in the new coordinate system.
You can input the coordinates in the "old" system in the left part of the dialog, and the "new" coordinates in the right part. The button *Define reference system* will create a new UCS for you as desired.

You can simply copy the coordinates of a point (as below written) and paste them with “Paste from clipboard” button filling the empty spaces.

P25 *1511.342* *948.368* *117.280*



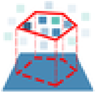
For more sophisticated methods of georeferentiation, see also Georeferentiation.

This tool creates a new UCS, based on the previous one and translated by an offset defined by the vector having the two points as extremes.

Selection & editing

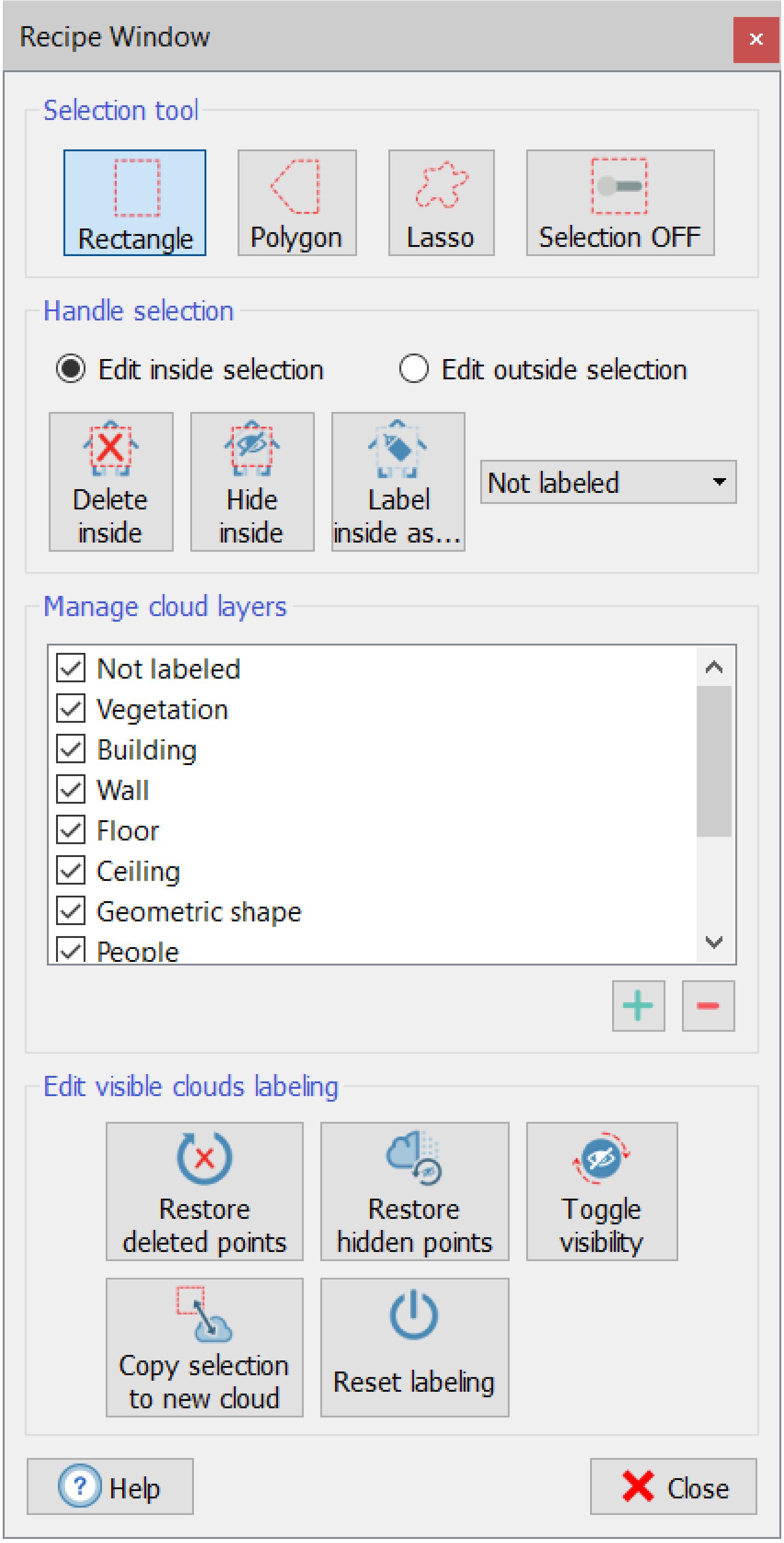
This category of tools includes functions able to select portions of point clouds, and manage them. These tools are grouped in two main families:

Select points
To select portion of point clouds, labeling and editing them.



Points selection with polyline
To select points by using a 3D polyline.

Select points



In this section you can learn how to select portions of point clouds to be:

- labeled
- deleted
- hidden
- copied to a new cloud

Deleting and hiding operations can be undone.

Selection Tool

*Switch between navigation (*Selection OFF*) and selection mode (*Selection ON*) by pressing the spacebar.*

Remember that the navigation in the 3D view (changing of point of view) can only be done in Selection OFF mode.

The pick mode is the way to interact with the 3D data displayed in the rendering window.
The *Selection* command helps the user to sample points in the current visible frustum from the loaded point clouds. Both 3D information and color are sampled. The shape of the region can be chosen with the selection mode menu.

Rectangle [Ctrl+1]
To make a rectangular selection of the point clouds in the current view.
Press Left Mouse Button (LMB) for the first point and keep it pressed while dragging the mouse to the second point, then release the LMB.

Polygon [Ctrl+2]
To make a polygonal selection of the point clouds in the current view.
click LMB for each point of the polygon. Close the polygon by double clicking the LMB.

Lasso [Ctrl+3]
To make a free hand selection in the point cloud view.
Press Left Mouse Button (LMB) to start the polyline and drag the mouse to draw, then release the LMB to close the polyline.

Complex selections can be performed by holding down special keys:

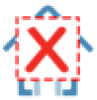
- Union: [*Shift*]
- Difference: [*Alt*]
- Intersection: [*Shift+Alt*]

When starting use the Polygon *mode and switching (without closing the perimeter) to the* Rectangle *or* Lasso *mode, a particular selection given by the combination of the two modes is available.*

Handle selection

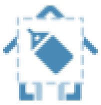
There are two ways to manage points just selected:

- Edit **inside** selection



Delete inside [Ctrl+Del]
The points that fall inside the selected region are deleted from the view. They can be undeleted at a later moment.

Hide inside [Ctrl+H]
The points that fall inside the selected region are hidden from the view. They can be unhide at a later moment.



Label inside as...
The points that fall inside the selected region are labeled according to the layer selected on the drop-down list to the right.

- Edit **outside** selection

Delete outside [Ctrl+Shift+Del]
The points that fall outside the selected region are deleted, and can be undeleted at a later moment.

Hide outside [Ctrl+K]
The points that fall outside the selected region are hidden from the view. They can be unhide at a later moment.

Label outside as...
The points that fall outside the selected region are labeled according to the layer selected on the drop-down list to the right.

Manage cloud layers

All the points in the 3D view can have only one label (included the default one "Not labeled").
All the available cloud layers (labels) are listed in the *Manage cloud layers* box.
By checking/unchecking them, the points related on them, are visible (or not) in the 3D view.

By using the and commands it is possible to add a layer to the list or delete a selected layer from the list, respectively.

This management is useful for displaying only some desired levels (e.g. if you don't want to see "People")

Edit visible clouds labeling

Once the points are selected and labeled, differents operations can be done:

Restore deleted points [Ctrl+Shift+Del]
Restores the points deleted from the visible point clouds.
See also Restore Deleted Points to learn how to undelete points from 2D view.

Restore hidden points [Ctrl+R]
Restores all hidden points in the visible point clouds, setting them as visible.

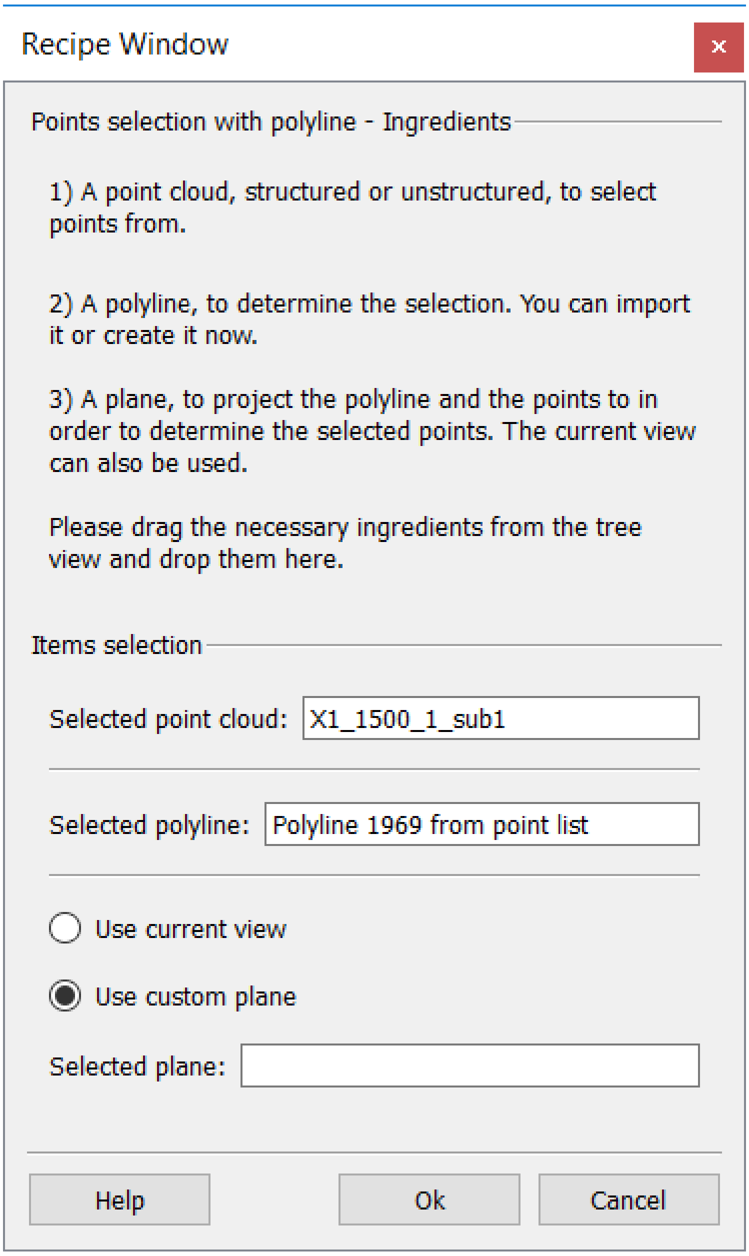
Toggle visibility [Ctrl+T]
Switches the hidden points back to visible and hides the visible points.

Copy selection to new cloud [Ctrl+Shift+C]
Copies the points that fall in the current video selection to a new point cloud.

Reset labeling [Ctrl+Shift+Del]
Resets the labeling applied to the points, which will be restored to "Not labeled".

During the standard pipeline of importing, preprocessing, registering and georeferencing of point clouds, you can edit one or more models by performing a manual removal of points (however a reversible operation) by the *Delete inside/outside* command. After cleaning the models, you can use the *Hide inside/outside* tool to highlight the regions to be labeled. A Toggle visibility button is also used to invert the visibility of the selected regions.

Point selection with polyline



This is a tool for selecting points using a 3D shape, in particular a polyline.

Drag the point cloud to be used as source of your selection on the recipe window. Then, drag the polyline you want to use as contour to select points from your cloud. Then, you need to specify a planar reference. This can be the *current view* or a *custom plane.*

The procedure works as follows: the polyline is projected on the planar reference you chose. If the polyline is not closed, it is automatically closed by connecting the last point to the first. Then, the polyline projected on the plane delimits a 2D closed region. The points of your cloud are also projected on the plane: the ones falling inside the projected polyline are marked as selected, the ones falling outside are marked as not selected.

If no points were selected, Reconstructor gives a warning message and stops the procedure. If any points were selected, Reconstructor creates a new unstructured point cloud called *Selection of <input-point-cloud-name>* and stores it as a new item in the project.

Cloud Processing

This category of tools includes functions dedicated to work with point clouds.
By applying these functions, other processes and further results are enabled.
Most of these tools work on any point cloud, some of them work only on grid point clouds (tripod scans).


Pre-process point clouds
Applies a set of standard algorithms to process point cloud data.


Merge clouds into one
Sums together a set of point clouds in one unstructured point cloud. A sampling factor can be defined.


Merge clouds and level3D density
Clusters together a set of point clouds in one unstructured point cloud. The final cloud density and sampling strategy can be defined.


Resample
Resamples the point clouds given a user defined subsample factor.


Simplify points (tripod scan only)
Makes an unstructured point cloud from the structured selected cloud and creates an hyperlink between the output and the original grid.


Extract edges (tripod scan only)
Extracts the edges as polylines from a grid point cloud. The selected point cloud must be preprocessed running the edge detection filter.

Fill holes (tripod scan only)
Tries to fill small gaps between points

Hide black points
Deletes all the black points (RGB [0, 0, 0]).


Edit 2D (tripod scan only)
Shows a structured point cloud in a 2D view editor.

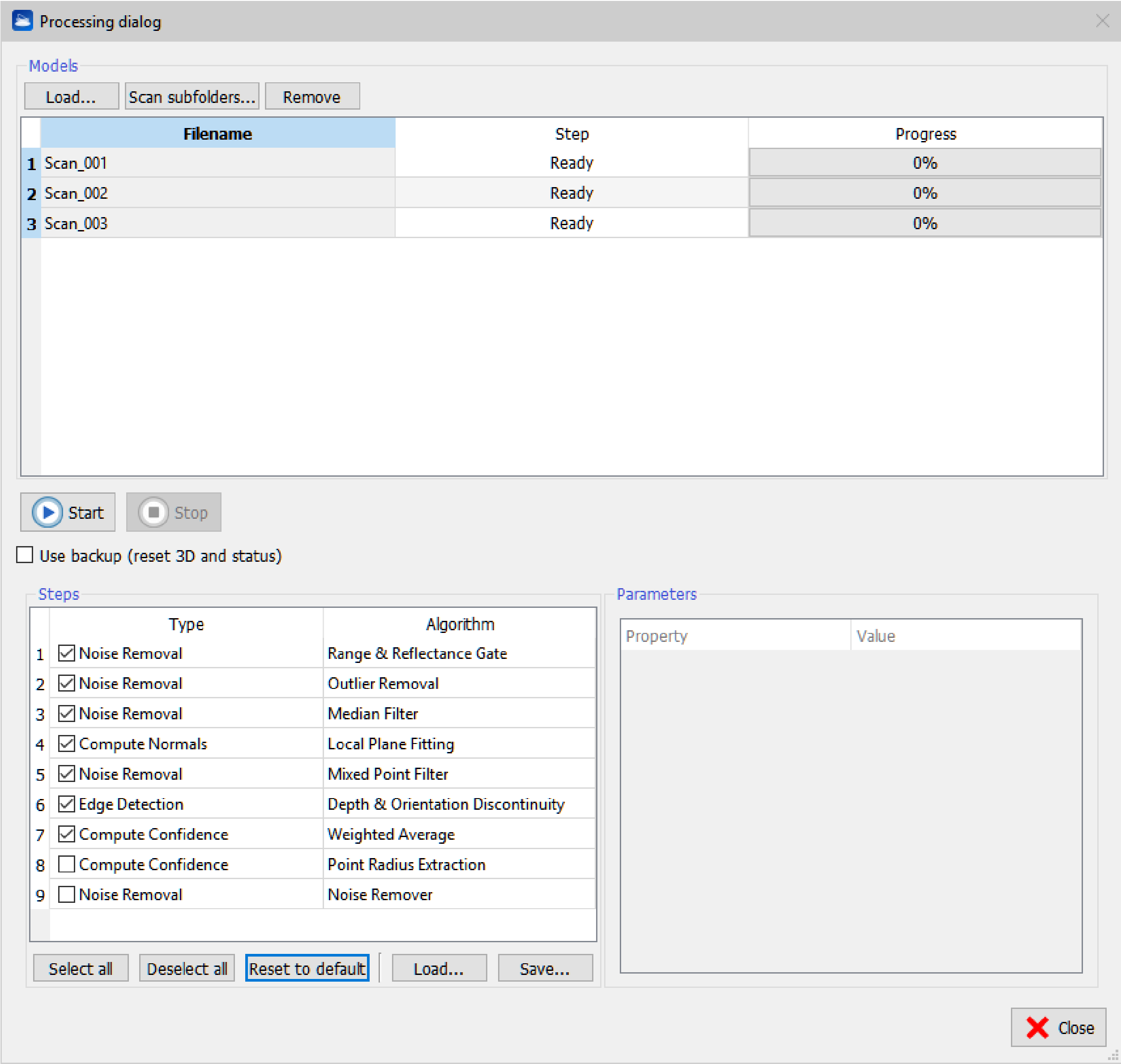
Remove duplicate points
Removes copies of points with the same coordinates inside a given point cloud.

Invert normals
Inverts the normals direction a selected point cloud.


Restore deleted points
Restores the points deleted from the point cloud.

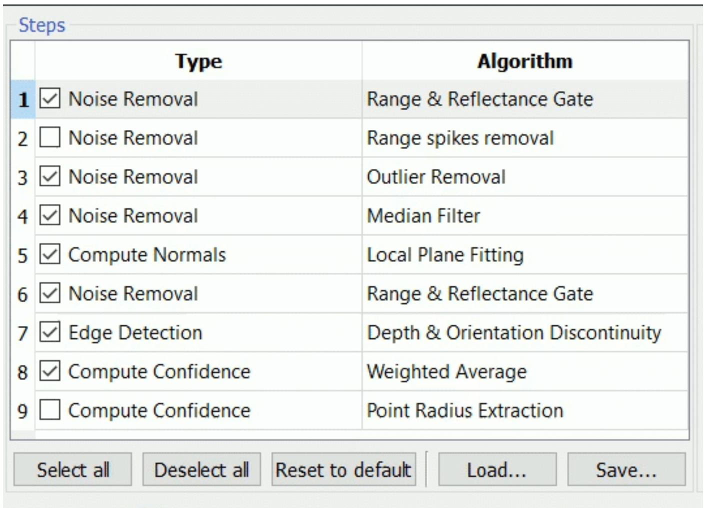
Restore raw data
Removes any changes resulting from pre-processing or editing from selected point clouds. The item position will not be changed.

Pre-processing Point Clouds



During preprocessing Reconstructor® applies a set of algorithms to the range scans, which extract information that is needed during further processing of the data. Press **Load** to add an external grid point cloud or **Scan subfolders** to find automatically all grids in a folder and its subfolders.
Press **Remove** to remove undesired grids. It is possible to process grids present in the current project by selecting *Pre-process* from the context menu of the grid item in the Project Window. In this case, when the processing is finished, the grid in unloaded from the project to free memory. Please force the reloading to refresh the rendering. Press **Start** to run the processing. This plug-in creates a separate thread for each grid so if multiple CPUs are available the processing is sped up proportionally.
All *steps* are applied consecutively to the selected grid by pressing the *Select All button*.
Reconstructor® never overwrites the original data. The first time the pre-processing begins, a backup of the data is created. By default the pre-processing pipeline starts from the data in its current state. If Use backup is checked, the pre-processing reads the backup data.

A *step* algorithm can be changed by clicking the algorithm name, a combo box of available algorithm is shown. A *step* can be skipped by unchecking it.
The drop-down menu appears if clicking the *Type* item and the arrow on the right:



Noise Removal

Range & Reflectance Gate

Min/Max range [m]: lower/upper threshold of the range. All pixels outside this range will be filtered.
Min/Max reflectance [0÷1]: all measurements that have an intensity value outside the given interval will be filtered.

Outlier Removal

Outlier removal is a Noise Removal filter that permits to delete the lonely points outside a detected surface, following this concept:
every point is investigated, and the N. points (i.e. Neighborhood Size 24) closer to it in the 3D space are detected. Also these points have N. points closer to them. If some point has neighbors, but these neighbors haven't it as a neighbor (so there aren't Incoming Edges), it means that this is an outlier.
The min. outlier likelihood is the probability that a point is an outlier: [1- (N. of Incoming Edges)/(Neighborhood Size)]
Lower is this value higher the number of outlier points, starting from isolated points, until groups and edge points.
Neighborhood size: number of points closest to the current point. The minimum settable value is 11, while the maximum value is 99.
Min. outlier likelihood [0÷1]: probability that a point is an outlier = [1- (N. of Incoming Edges)/(Neighborhood size)]

Mixed Point Filter

Useful as a relative depth discontinuity filter.
Min incident angle [deg]: if the line of sight from the origin of the grid to the point has incident angle to the local surface less than this value, the point is filtered.
Mask border [pix]: the kernel of the filter is a square mask (of side 2·border+1) centered at the current pixel.
Please note that this filter works only on grid point clouds.

Median Filter

Mask border [pix]: the kernel of the filter is a square mask (of side 2·border+1) centered at the current pixel.
Please note that this filter works only on grid point clouds.

Mask Filter

Loading a 2D image file (*Mask file*) in which some points are colored in black (RGB = 0,0,0), the corresponding points in the 3D point cloud will be deleted.

The 2D image must derive from the 2D grid (look at *Save Image* in Edit 2D), then processed in a photo editing software.

Range Spikes Removal

This filter attempts to remove range discontinuities by looking at the average range of a neighborhood of each point. Range discontinuity threshold is set to 200m and can be changed. Neighborhood size is set to a 10 x 10 square around any given point and can be changed by editing the "mask size parameter". Please note that this filter works only on grid point clouds.

Statistical outliers remover - SOR

This filter firstly computes the average distance of each point to its neighbors (where the number of nearest neighbors is defined by the *Neighborhood size*). After that it rejects the points that are farther than the average distance plus a number of times (*Standard deviation threshold*) the standard deviation.

Compute Normals

Local Plane Fitting

Computes the local surface tangent plane for each point, based on the neighborhood of the pixel.
Mask border [pix]: the kernel of the filter is a square mask (of side 2·border+1) centered at the current pixel.
Neighborhood size [pix]: pixels frame close to the current pixel.
Compute also curvature: set TRUE to compute it.
Adjust normals' verse in handheld scans: set TRUE if processing handheld scans.

Edge Detection

Depth & Orientation Discontinuity

Computes geometrically significant line features from the point cloud data. Two types of edges are extracted:

- Depth discontinuities (or jump edges) that occur where the scanner hits an occlusion and therefore the measured range jumps from a foreground to a background value.
- Orientation discontinuities (or crease edges) that occur where the object has a sudden change of its surface orientation.

The extracted edges are stored as a bit flag for each point.
Mask border [pix]: the kernel of the filter is a square mask (of side 2·border+1) centered at the current pixel.
Min depth discontinuity to flag [m]: absolute depth discontinuity to filter.
Min orientation discontinuity to flag [deg]: the current point is marked if its normal differs from the adjacent ones of at least this angle.

Compute Confidence

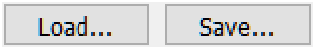
Weighted Average

Computes a confidence value for each measurement, which is a measure for the reliability of the given range measurement. The accuracy does not only depend on the type of scanner used, but amongst others also on the following factors:

- The incident angle between the laser beam and the tangent plane of the target
- The distance to the target
- The material of the object and therefore the intensity of the reflected signal

The confidence value is computed as a weighted sum of the surface normal, the range value and the reflectance value.
Min/Max range [m]: the range is weighted by normalizing it within this interval.
Scale factor: the confidence value calculated for each pixel is multiplied with this scale. Thus the user has the possibility to decrease or increase the weight for a given scan manually, for example because a scanner with higher accuracy has been used during acquisition.

Weight of range/reflectance/inclination: modify the weight as desired.



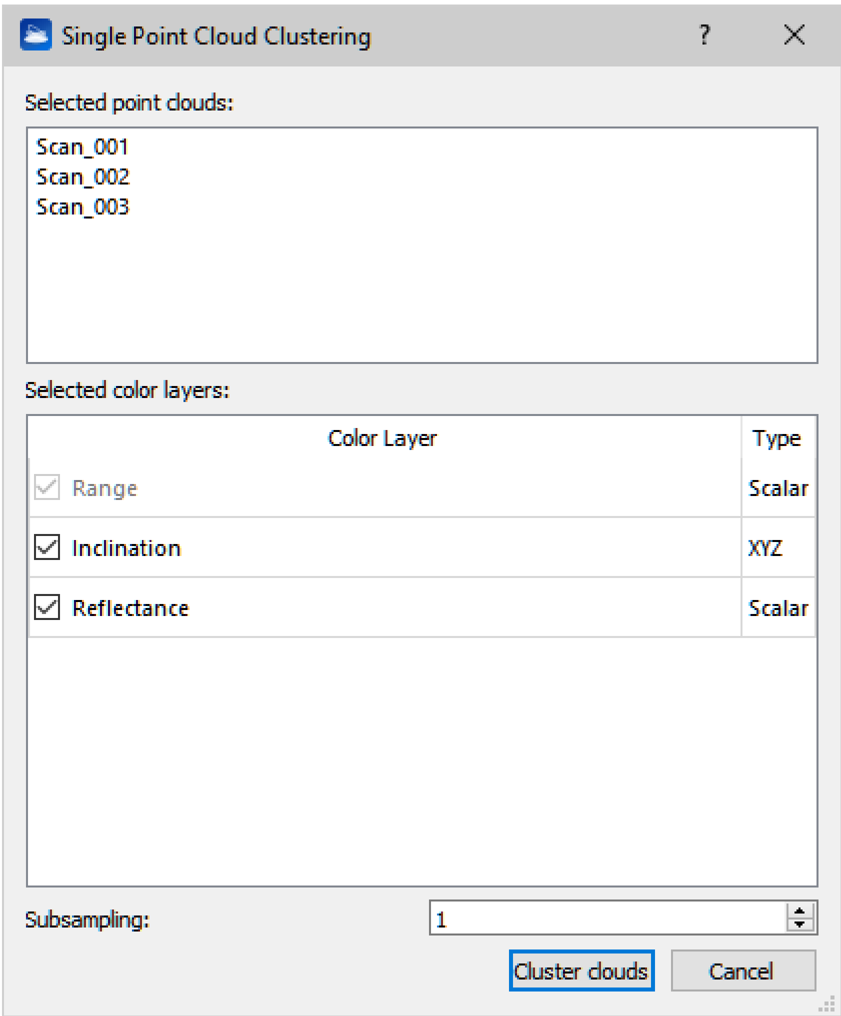
Save and *Load* your custom settings for specific surveys and elaborations.

Point Radius Extraction

The Point Radius Extraction define the amount of surface that every point can represent. The Max. Radius is the maximum radius of the aforementioned area (it's suggested to not change this value). The result we'll be useful to splat rendering.

Merge Clouds Into One

(Single Point Cloud Clustering dialog)



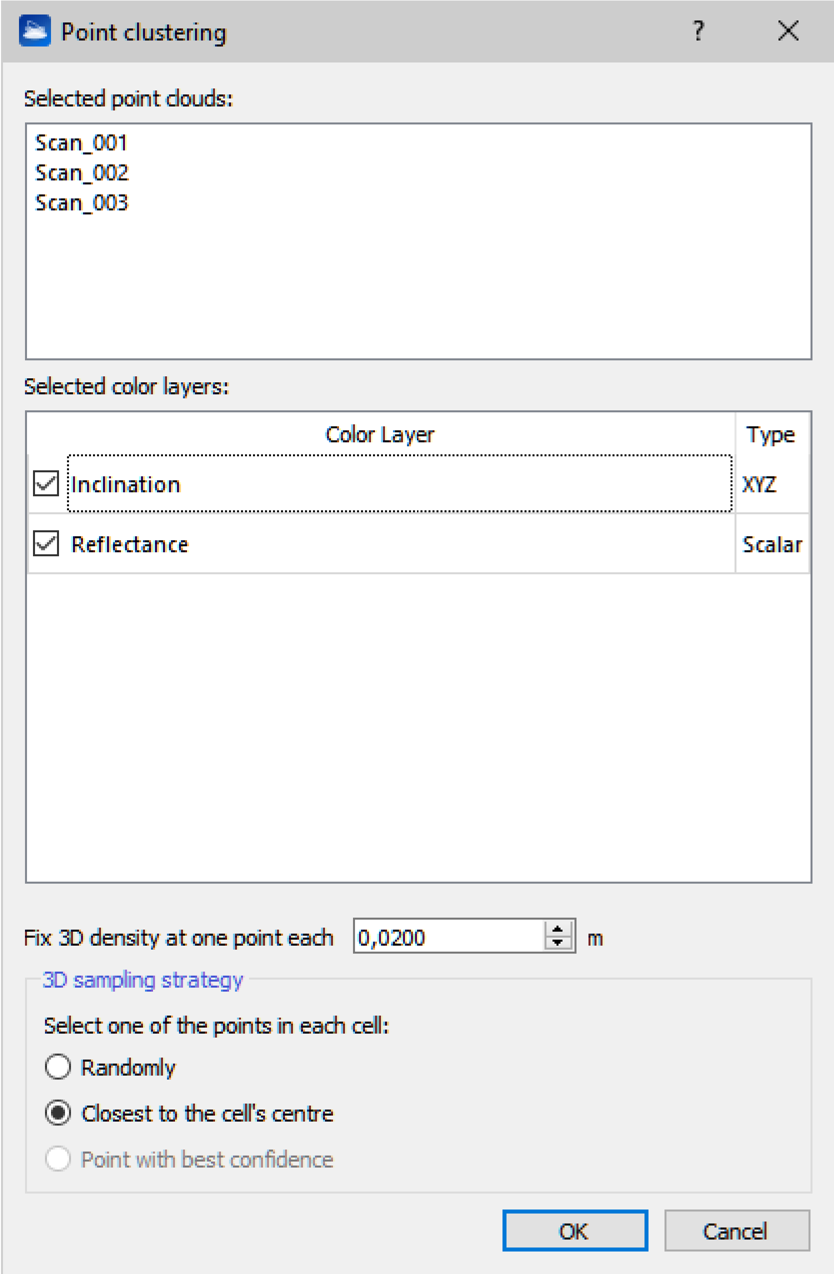
This dialog allows to lump together in an unstructured point cloud an arbitrary set of point clouds, structured or not. If you want to cluster clouds excluding duplicated or unneeded points, you may want to take a look at Level 3D density of clouds.

The selected clouds appear in the top list of the dialog. Below, the common color layers across the clouds are found. The cloud that results from the clustering will contain only the color layers that are present in all input clouds and that are chosen. Point normals will be transformed so that they will be correct in the resulting cloud.

Below in the dialog, you can specify a subsampling step if you do not wish to include all points of the input clouds in a new cloud.

The clustering procedure will efficiently exploit all CPU cores of your PC.

Merge clouds and level3D density



This dialog allows to lump together in an unstructured point cloud an arbitrary set of point clouds, structured or not. The resulting cloud, however, will not contain all points from the input clouds, but only those needed to guarantee a fixed 3D density of the points.

Regularize the density of your clouds

Laser scanner data can have quite irregular 3D densities. For example, objects very close to the laser scanner may be sampled at one point each half millimeter, but objects far away from the scanner may be sampled at one point each 30/40 cm. This tool allows you to regularize the 3D density of your clouds. The resulting regularized cloud can then be used for meshing or for exporting in CAD tools for further analyses.

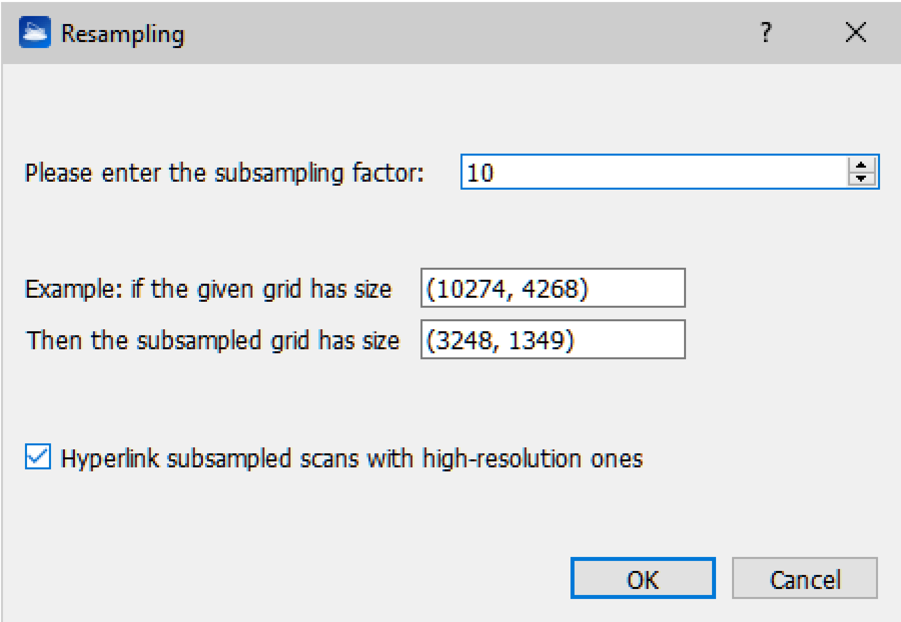
The selected clouds appear in the top list of the dialog. Below, the common color layers across the clouds are found. The cloud that results from the clustering will contain only the color layers that are present in all input clouds and that are chosen. Point normals will be transformed so that they will be correct in the resulting cloud.

Options

In the lower half of the dialog, some options are present. The first is of course the desired 3D density, expressed in minimum point-to-point distance. Then, you can also select the strategy for choosing the representative point of each 3D cell (defined by edges equal to the set 3D density - center point-to-point distance):

- *Randomly*: the point will be randomly chosen between the points within each 3D cell volume
- *Closest to the cell's centre*: the point will be randomly chosen between the points within each 3D cell volume
- *Point with best confidence*: the points with higher confidence will be chosen

Resample



This tool allows you to resample point clouds. It works on any set of grid point clouds or unstructured point clouds.

On top of the dialog you are asked to input the subsampling factor *s*. A new point cloud will be created taking one point each *s* from the original cloud. If you are resampling grid point clouds, the resampling operation will take into account the structure of the grid. Therefore, the grid structure will be divided in cells with size square_root(*s*) by square_root(*s*), and one point will be taken for each cell. To clarify this, in the grid point cloud case the dialog shows you a message with the original and the resulting grid size, as in the figure above.

Optionally you can hyperlink the subsampled scans with the higher resolution ones. This allows you to work with a lot of data by doing operation on the light versions of the data.

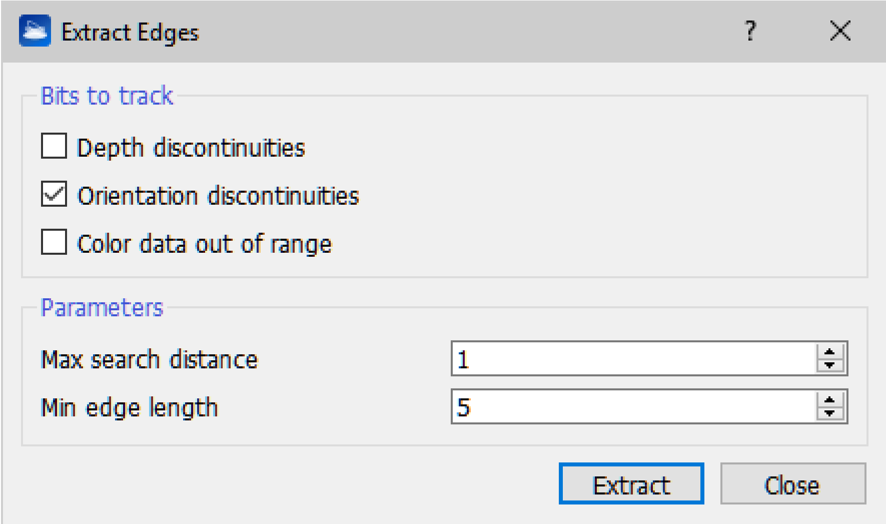
Simplify Points

Input data: at least one grid point cloud must be selected.
Output: a new unstructured point cloud is created for each structured cloud in input.

This functions takes as input a set of structured point clouds. For each of them, it determines the most relevant points from a point of view of shape description, and it saves them into the new unstructured point cloud. These resulting clouds work as compact representations of the original structured ones. They can be hyperlinked to the original clouds, so that you can open the original data on demand.

The function works by creating a mesh in background of each input cloud, and by inserting into the resulting cloud the mesh's vertices.

Extract Edges



This tool allows you to extract the edges of a grid point cloud, in form of polylines. You can also activate this function from the top menu by clicking *Data management* > Cloud processing > *Extract edges (tripod scans)*, or by the context menu of any grid point cloud.

The selected point cloud must be pre-processed running the Edge Detection filter.

You can select which discontinuities in the 3D point cloud you want to mark as edges. These can be the *Depth discontinuities* (points that are neighboring in the grid structure but far away in 3D position), the *Orientation discontinuities* (points that are neighboring in the grid structure, close to each other in 3D position, but having normals very different), or both.

The parameters *Max search distance* and *Min edge length* define the way edges are extracted. *Max search distance* is expressed in pixels and refers to distances in the grid's structure (range image). *Max search distance* determines how far away in the grid structure two points belonging to the edges must be connected in the same edge. *Min edge length* means that all the edges composed by less points than this parameter will be considered as noise and discarded.

Fill Holes

Input data: exactly one grid point cloud must be selected.

This functions takes as input exactly one grid point cloud, and tries to replace any invalid point in the cloud with a value averaged from the point's neighborhood in the cloud's structure. After activating this tool, Reconstructor asks you the mask border M: the search neighborhood of each point is defined by the (2M + 1) * (2M + 1) points located around the given points in the cloud's structure.

This tool is useful to cover small points in a cloud's structure, in order to get more connected meshes using the multiresolution mesh.

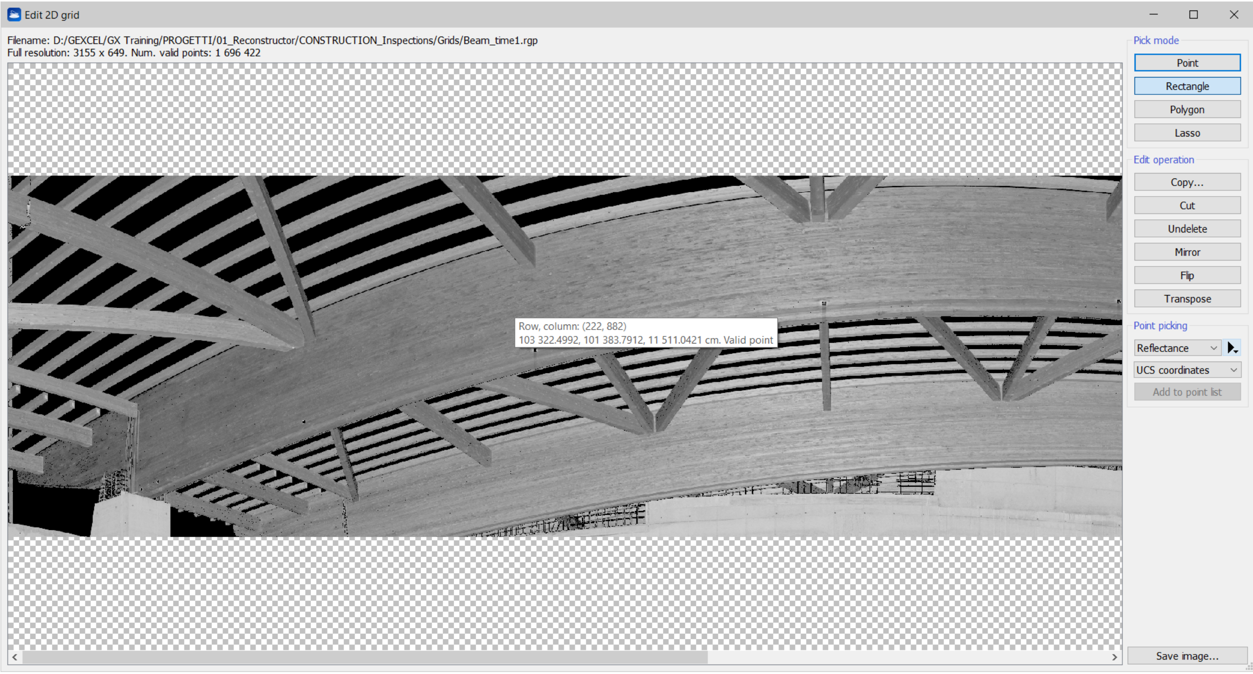
Hide Black Points

Input data: at least one structured or unstructured point cloud must be selected.

This functions takes as input a set of point clouds, for each of them takes the current color layer and hides all the points in the cloud that are colored in full black, i.e. RGB components [0 0 0].

The hidden points can be made visible again with the*Restore hidden points [Ctrl+R]* command in the Select points tool.

Edit 2D Grid Window



This window shows a grid point cloud in its 2D representation. Like in a geographic 2D map of the world, the grid point cloud is represented as an image where, for each point, the x coordinate is the yaw angle (longitude) and the y coordinate is the pitch angle (latitude) that the laser had while surveying that point.
In this window you can select, delete and undelete points with several functions.

To interact with the image

- Zoom in/out: Alt + mouse wheel
- Fit the image in the window: Alt + Middle Mouse Button (MMB)
- Reset to original size: Alt + Right Mouse Button (RMB)
- Translate the image: Shift + LMB drag
- Space bar: rotate image 90 degrees
- Show/hide zoom window: Z key
- Move zoom region inside global view: [Ctrl + LMB]
- While keeping pressed ...
  - Shift: to perform union of regions
  - Alt: to perform subtraction of regions
  - Shift+Alt: to perform intersection of regions

Pick mode

- **Point**: pick a point and save it in a point list if *Add to point list* button is pressed.
- **Rectangle**: press Left Mouse Button (LMB) for the first point and keep it pressed while dragging the mouse to the second point, then release the LMB
- **Polygon**: click LMB for each point of the polygon. Close the polygon by double clicking the LMB
- **Lasso**: press Left Mouse Button (LMB) to start the polyline and drag the mouse to draw,

then release the LMB to close the polyline

Edit operation

- **Copy**: create a new grid point cloud from the selected region
- **Cut**: delete the selected region
- **Undelete**: undelete the points inside the selected region
- **Mirror**: inverts the grid along the width. Only the ordering of the grid is changed, the 3D points are untouched
- **Flip**: inverts the grid along the height. Only the ordering of the grid is changed, the 3D points are untouched
- **Transpose**: transposes the grid. Only the ordering of the grid is changed, the 3D points are untouched.

Point picking

- **Color layer**: display the grid with the selected color layer. Press the right arrow to show a menu of available commands for the current color type
- **Histogram**: press the black arrow to select the histogram tool that allows to optimize the contrast by histogram stretching (available only if the model is loaded in memory and if the current color type is 1f or 1d, i.e. a high-dynamic single-channel point color).
- **Object/UCS/world coordinates**: display the coordinates of the point into the desired reference system (from object pose, in the current UCS and in the Main UCS).
- **Add to point list**: add the picked points to the project's point list.
- **Save image**: save the current image to file.

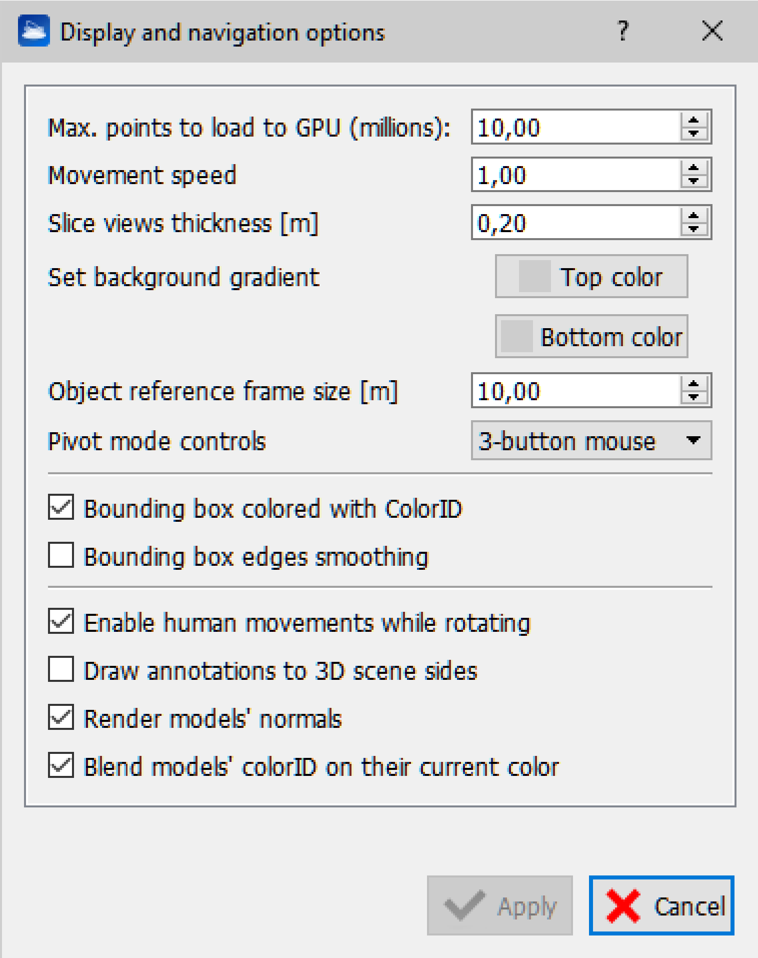
Remove Duplicate Points

Input data: at least one unstructured point cloud must be selected.
This functions takes as input a set of point clouds and, for each given cloud, it invalidates any point that has exactly the same coordinates of another point in the cloud.
Since JRC Reconstructor® 3.2, this function is automatically applied to all point clouds while importing them. Therefore, you should not need to use it. However, you can run it on clouds imported with previous Reconstructor® versions. After its conclusion, this function prints to the Log window the number of removed points per cloud.
Once removed, the duplicated points cannot be restored in any way.

Many laser scanners write raw data files with a lot of points in (0, 0, 0) that, if not removed, risk to undermine further processes like filtering or mesh creation. When importing clouds into Point R^3, all duplicate points must be removed.

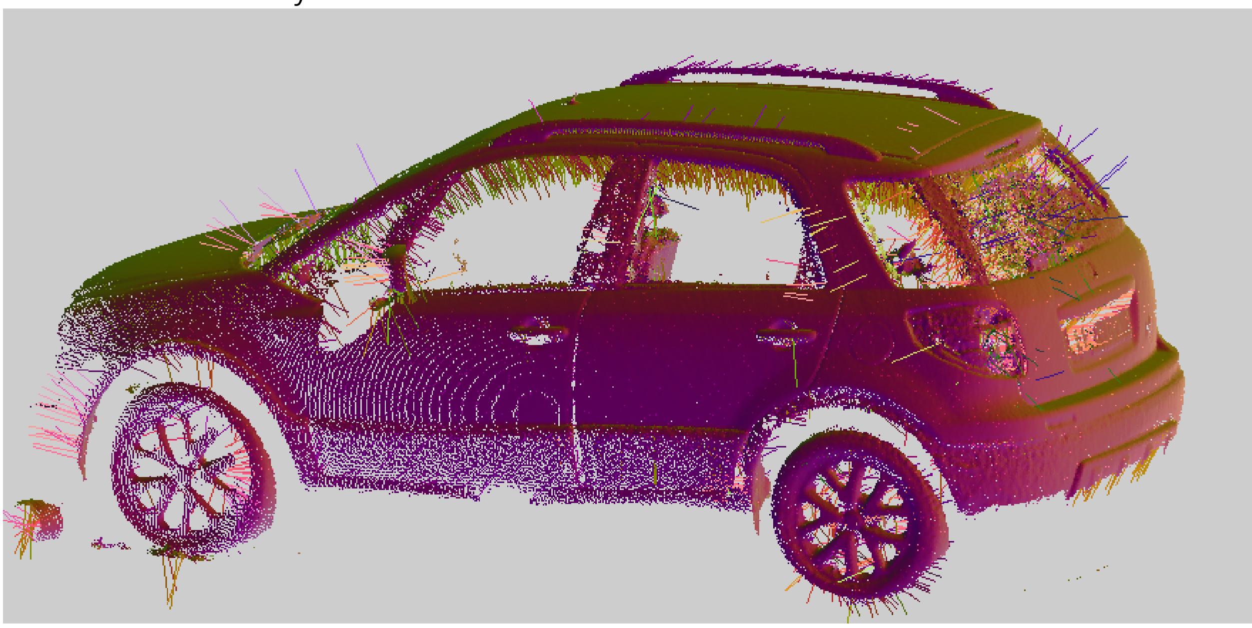
Invert Normals

Sometimes unstructured clouds present difficulties in calculating normals.
When the normals have opposite direction to the correct one this filter allows to flip them and solve the problem.

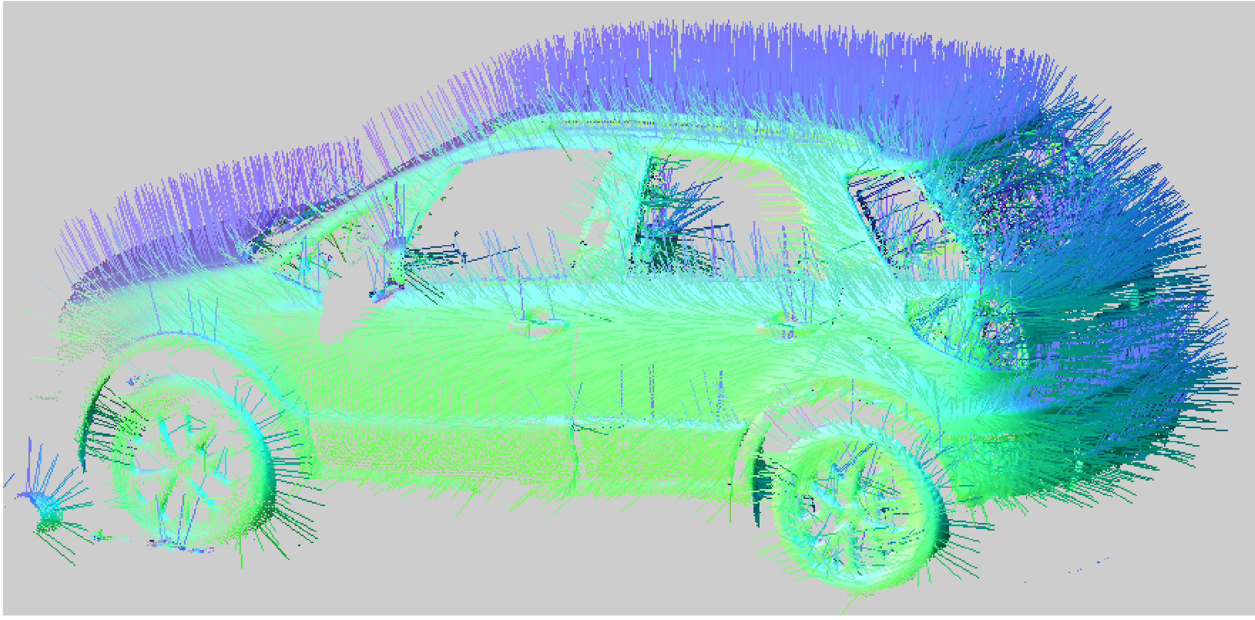


To turn on the "hairy view" open the *Display and navigation Options* window and check *Render models' normals*.

Normals on the contrary:

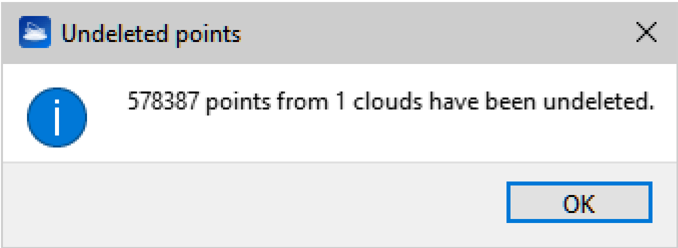


Correct normals after the *Invert Normals* command:

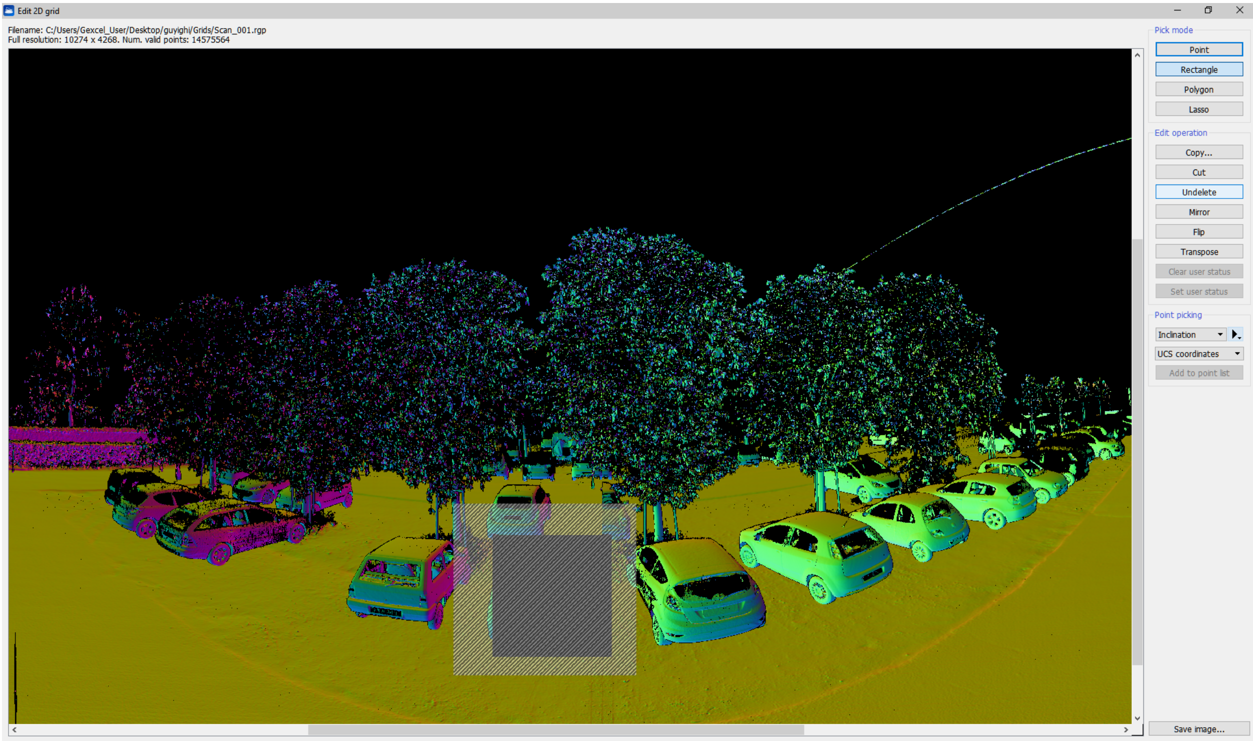


Restore Deleted Points

Sometimes happens that any point are deleted for mistake.
In Reconstructor® there's a method to recover deleted points, even in different times, valid for both Grid and Unstructured point clouds.
Restore Deleted Points (also available in the contextual menu of the point cloud) permits to undelete **all** points previously deleted.



If you want to undelete only the points in a defined area, the 2D edit it's the right way. It runs only for Grid point clouds.
Select a region containing the portion before occupied by the deleted points using a pick mode and then click on *Undelete* command.



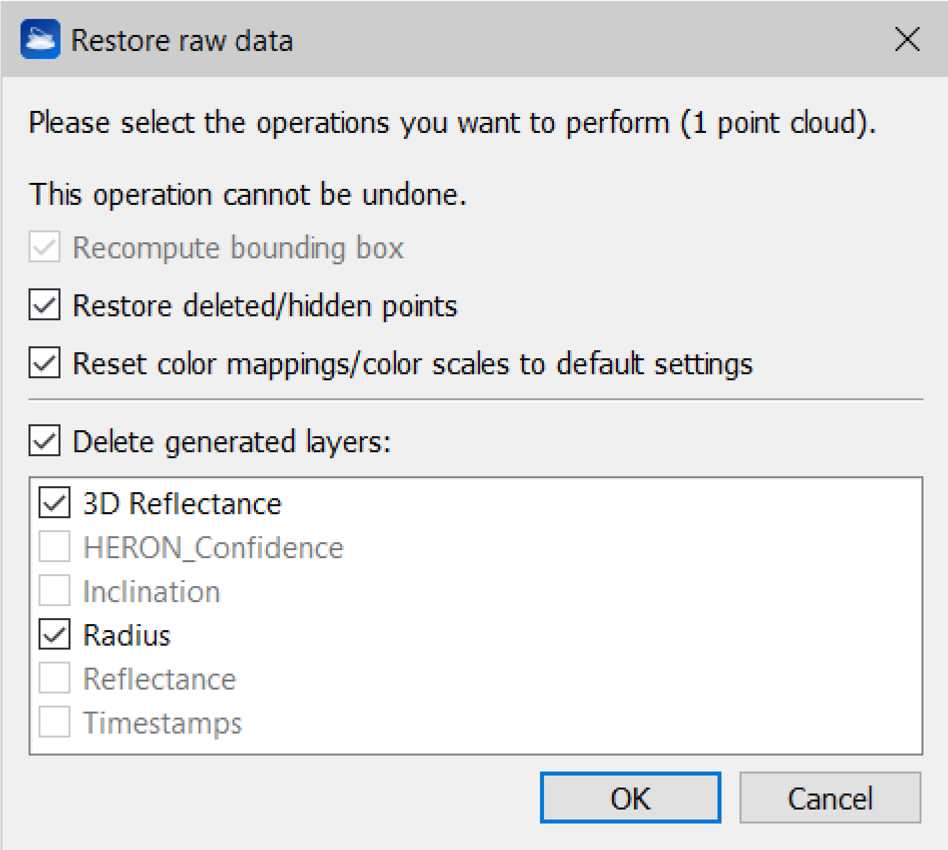
See also Edit 2D Grid.

Restore Raw Data

This functions takes as input a set of point clouds and undoes any operation of preprocessing, deletion and editing that may have been performed on the clouds.
Its main usefulness is to guarantee that you never lose data: if you are not happy with an editing operation or with the result of a filter, you can always go back to the clouds' raw data, as if they were just imported.
At least one point cloud must be selected as input data.

When applying this function:

- Points invalidated by the range and reflectance gate are valid again
- Points invalidated by the outlier remover filter are valid again
- Points invalidated by the mixed point filter are valid again
- The effect of any application of the median filter is undone
- Flags of points as depth or orientation discontinuities are deleted



If *Restore delete/hidden points* is flagged all deleted points are reset and all hidden points are visible again.

If *Reset color mappings/color scales to default settings* is flagged all the color scales of the selected point clouds are reset.

This operation can be used to recover some hard disk space. If necessary, it is then possible to re-modify the colors mapping settings with little effort.

A list of color layers available for the selected clouds is displayed in the second part of the dialog. Here it is possible to flag the color layers that you want to delete.

Between them two particular layers need attention:

- *Reflectance* layer cannot be delete, since it is usually included in the raw data
- *Inclination* layer can be deleted only if generated by standard Reconstructor's pre-processing.

The Restore raw data process is non-reversible, so be careful in using it.

Meshing

This category of tools comprehends functions dedicated to work with meshes:

- **Mesh creation** includes tools to extract different type of mesh from point clouds or list of points
- **Mesh manipulation** includes tools to edit and manage meshes, and convert them in point clouds.

Mesh creation

In this section you can first learn how to create triangle meshes.

Reconstructor® provides many typologies of mesh creation techniques.
Different number and type of point clouds can be used as a basis to create a mesh, depending on the meshing technique:

	On Grid Point Clouds	*On Unstructured Point Clouds*	*Suggested for...*
Multiresolution Mesh Fast meshing technique that give back light meshes that may have holes in some situations.	✓ (one or more)		To obtain a well defined and fast mesh from a single structured point cloud, with a good quality/computational time ratio
***Mesh* from predefined view** Relatively slow meshing technique that gives back convex meshes without holes. It's a view dependent, high defined mesh (each point is a vertex)	✓ (one)	✓ (one – single or clustered point cloud)	Useful for façades (using orthocamera) and tunneling (using cylindrical camera)
3D Mesh Approximative 3D meshing not view-dependent and taking as constraints the points' positions and orientations (normals)	✓ (one)	✓ (one – single or clustered point cloud)	Useful for convex surfaces.
Topographic Mesh The algorithm designed for DTM models. It gives back a watertight Fast meshing, light, smoothed mesh useful for isolines and volumes calculation.	✓ (one or more)	✓ (one or more)	Useful for land survey and mining
Mesh point list Two methods to transform a list of point into a mesh	✓	✓	To create simple meshes

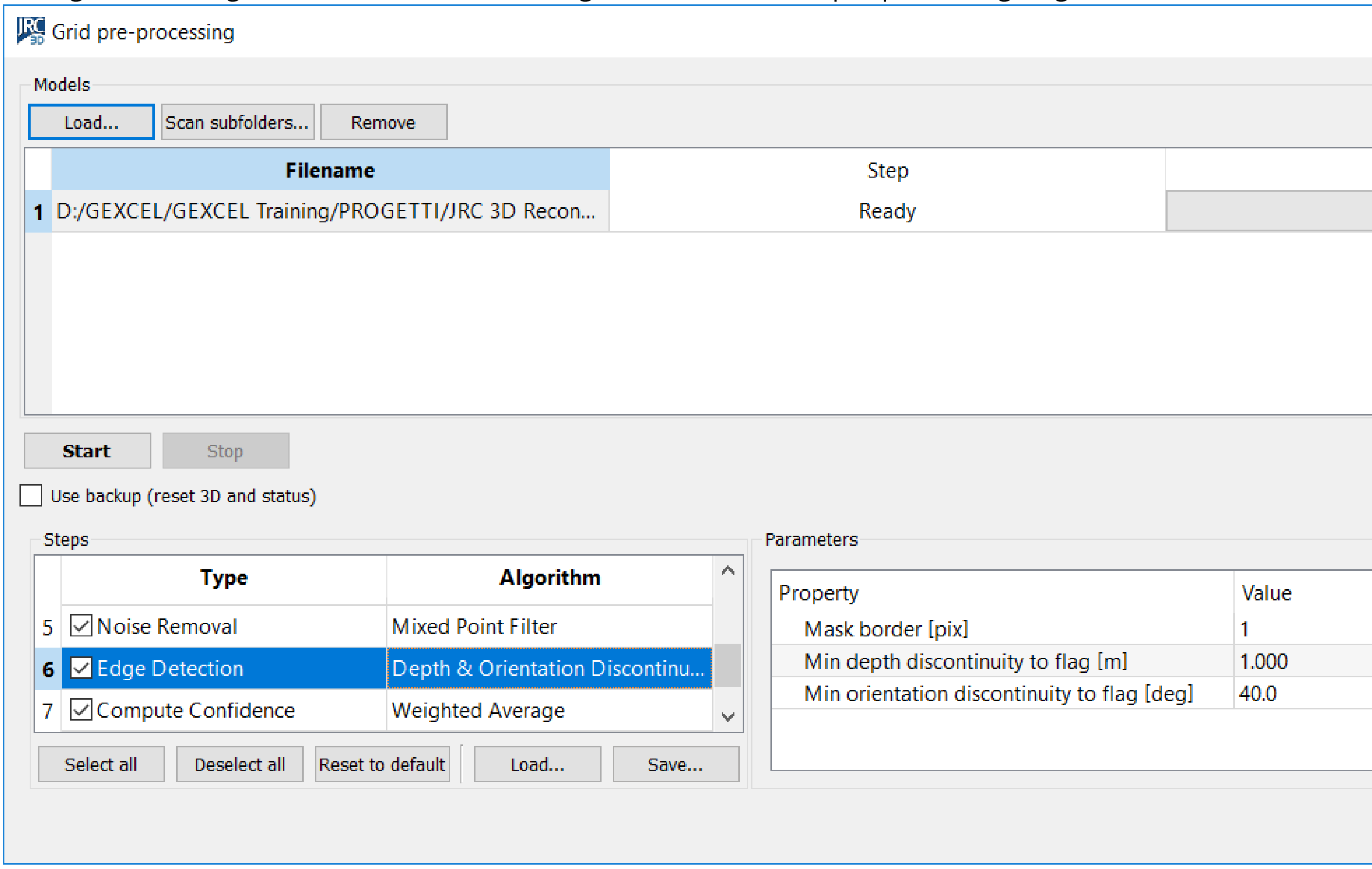
Multiresolution Meshing

Multiresolution Meshing is possible only with Grid point clouds. If you want to generate a mesh from an Unstructured point cloud, first transform it in a Grid point cloud by using the Virtual scan tool.

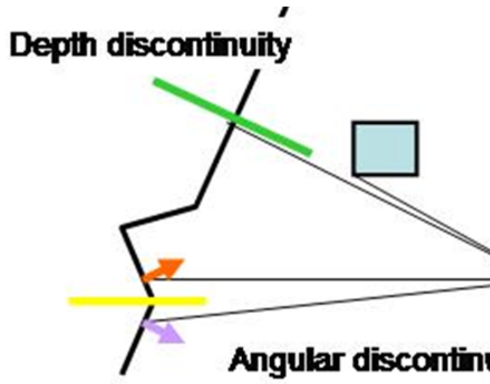
NOTE

Before meshing you need to Preprocess the Grid.

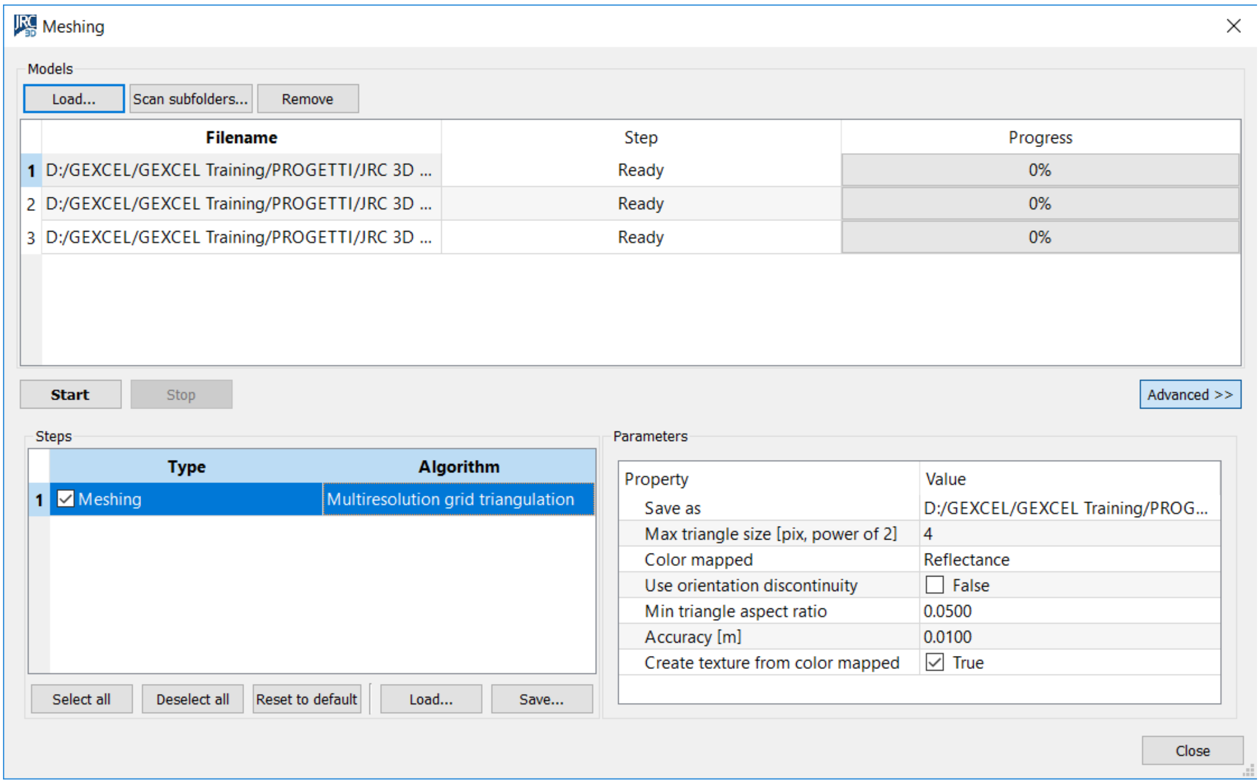
During the meshing the software will use the Edges calculated at the pre-processing stage:



- *Angular discontinuity* : set the angle [deg]; the higher the angle value, the higher the detail of the calculated edges. Close to the angular discontinuity you will have more detailed triangles
- *Depth discontinuity*: set the minimum distance [m] two points have to be considered as belonging to two separate objects→ no triangles are created between two far points. The lower the distance value the more holes are filled.



The Multiresolution Mesh tool is accessible from *Tools*→*Meshing*→*Multiresolution Mesh*, from the corresponding button in the top toolbar, or from the context menu of grid items in the project window.



- Press **Load** to add an external grid point cloud or **Scan subfolders** to find automatically all grids in a folder and its subfolders. Press **Remove** to remove undesired grids.

It is possible to process grids present in the current project by selecting Meshing from the context menu of the grid item in the Project Window. In this case, when the processing is finished, the grid is unloaded from the project to free memory. Please force the reloading to refresh the rendering.

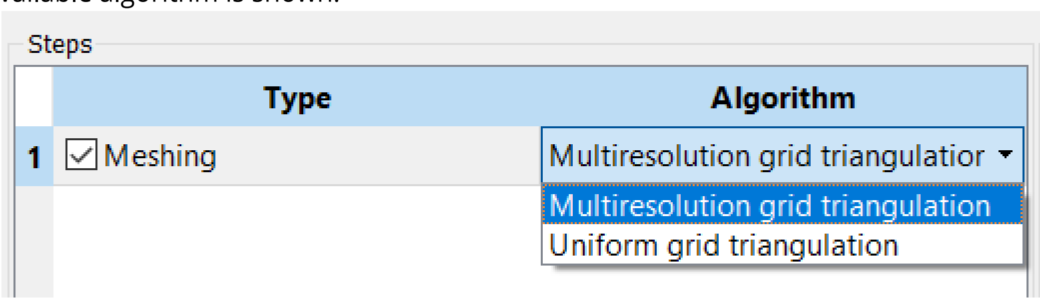
- Press **Start** to run the processing. This plug-in creates a separate thread for each grid so if multiple CPUs are available the processing is sped up proportionally.

The meshing process is applied consecutively to the selected grid by pressing the Select All button.

- To modify the meshing parameters press **Advanced**, then select one or more rows in the list of models with Ctrl or Shift. The parameters are updated for all the selected models.
- It is possible to **Save** the meshing parameter and **Load** them for other scans or projects or **Reset** them **to default** values.

Triangulation

The triangulation algorithm can be changed by clicking (three times) the algorithm name, so a combo box of available algorithm is shown.



Uniform

All the 3D valid points are connected with triangle meshes to build the range surface, without

simplification.
The algorithm checks if 2 points are divided by a depth discontinuity, in that case no “false” triangles are created.

Save as: check the file name of the resulting mesh.

Color mapped: select the color assigned to the vertexes of the mesh (e.g. Reflectance, Color, Inclination, Confidence, ...)

Multi-resolution

Simplified (lighter) triangle meshes are created in “flat” areas and dense triangle meshes are created in geometrical complex areas (close to edges or curvatures), satisfying an approximation error threshold.

Save as: check the file name of the resulting mesh.

Max triangle size [pix power of 2]: 2 $^{[\text{number of points}]}$, it’s the size of initial triangle; the value you can set is [number of points] you want to skip in the flat area. The higher the value, the more the mesh gets simplified. As example, a value of 4 means that a triangle will be created for a square wide 2^4=16 pixels. If the approximation error is too high, the triangle will be subdivide in smaller pieces.

Color mapped: select the color the color layer you want to map.

Use orientation discontinuity: thepre-processed orientation discontinuity is used to drive the subdivision process in order to preserve features such as edges. Set to TRUE to have more triangles around the edges.

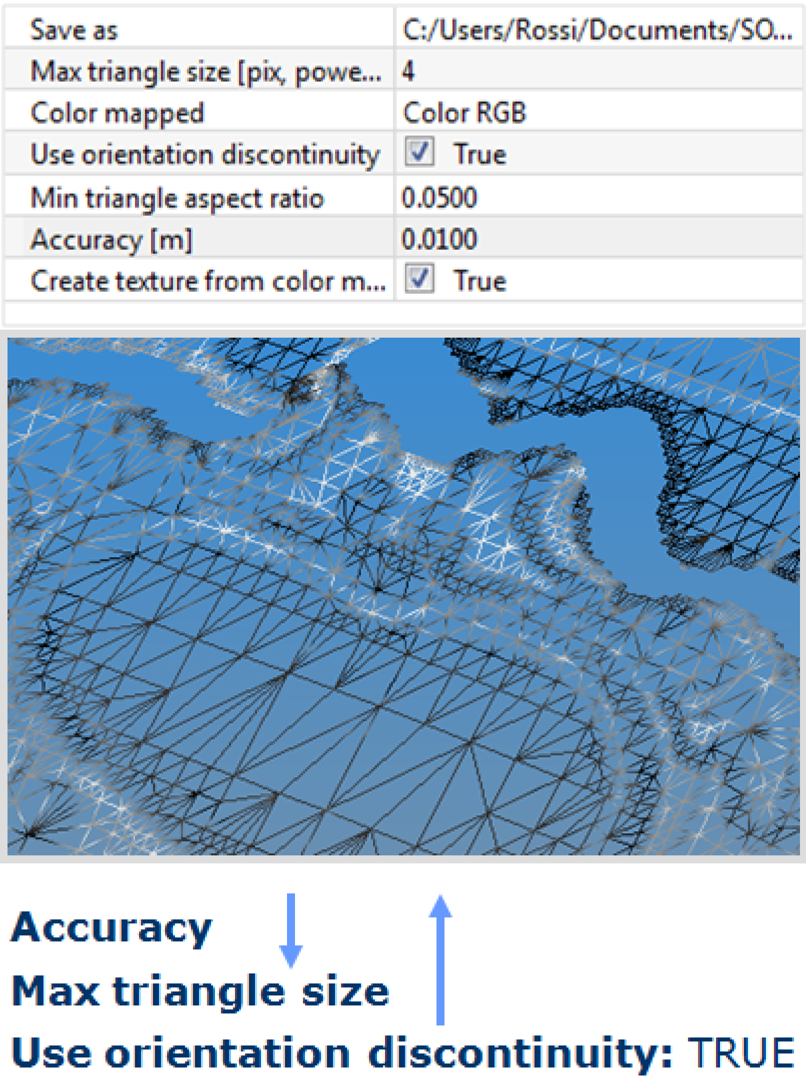
Min triangle aspect ratio: minimum accepted triangle aspect ratio. If the triangle is too thin it will be subdivided in smaller pieces. This test is performed only for the initial triangle.

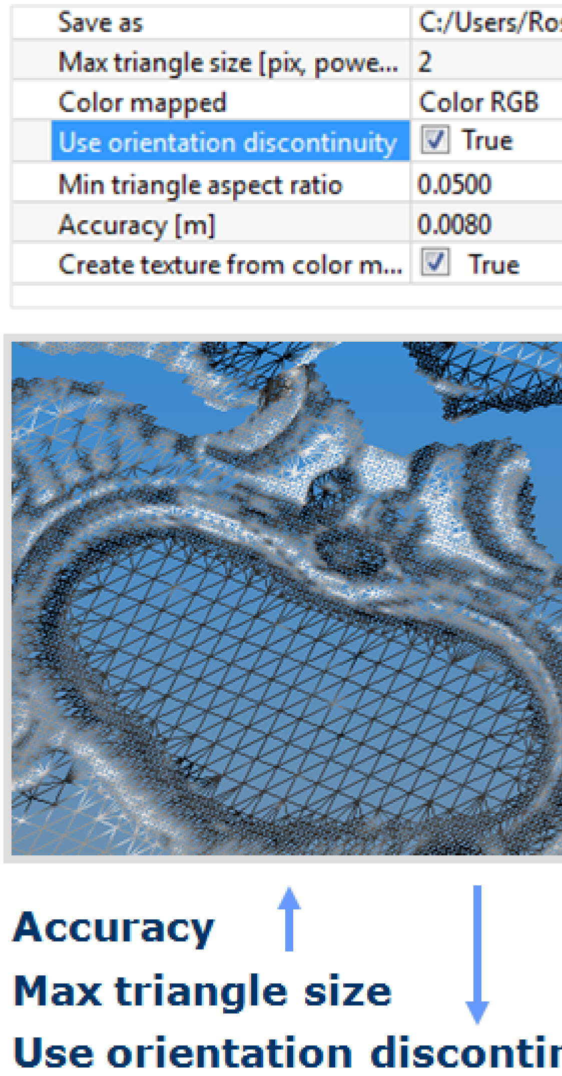
[Note: Aspect ratio of a triangle is the ratio of the inradius to its circumradius). The aspect ratio of a triangle lies between 0 and 0.5. For triangles with an angle near to zero, the aspect ratio is 0; for equilateral triangles, the aspect ratio is 0.5.]

Accuracy [m]: max approximation error of the mesh. The higher the value, the lesser triangles you have.

Create texture for color mapped: a texture is generated from the selected Color mapped on the mesh. Set to TRUE if you want the color layer not to lose resolution with the mesh simplification.

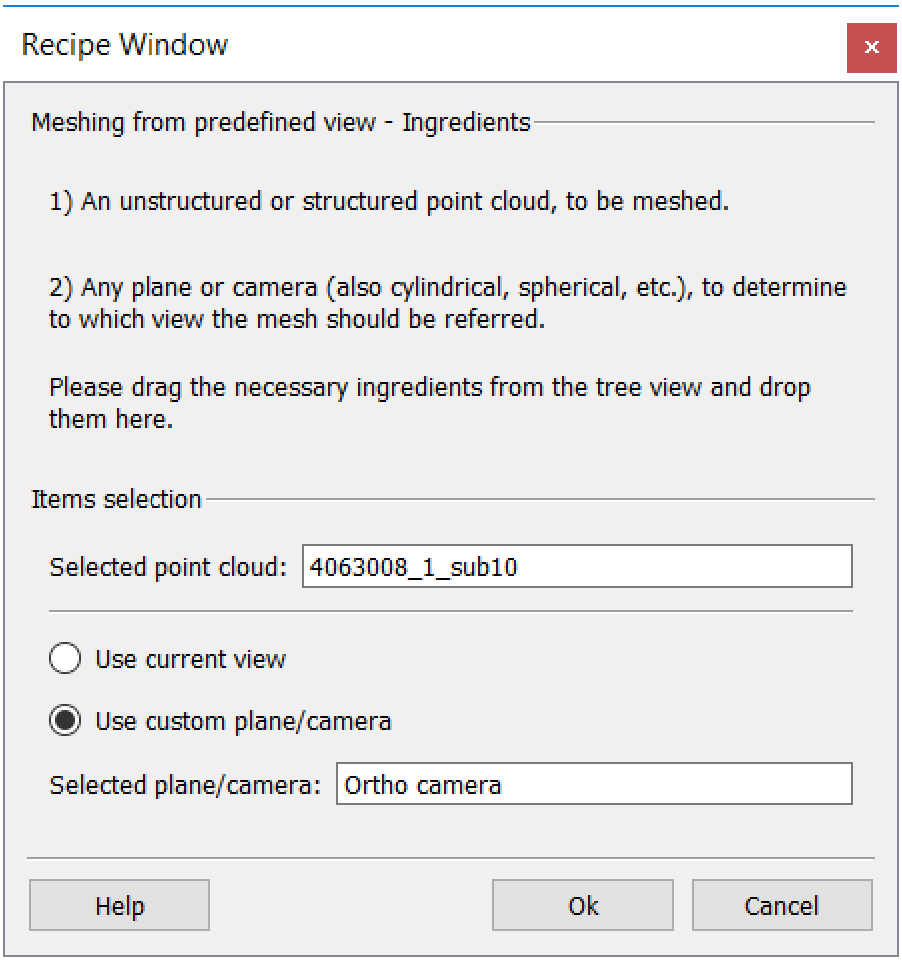
In the images below an example of the use of two different set of parameters is showed.





See also other Meshing techniques.

Meshing from predefined view

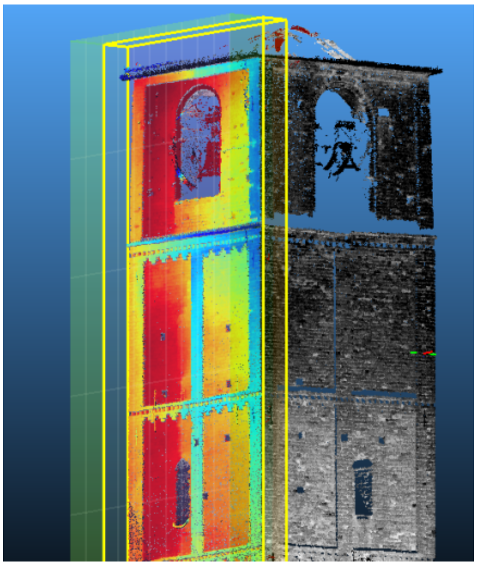


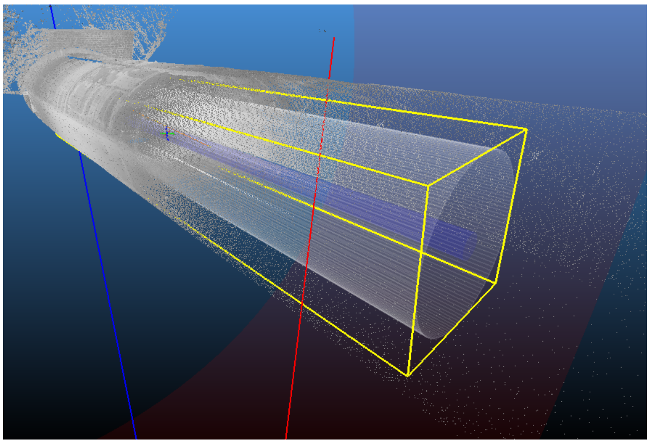
This is a meshing technique that provides watertight meshes. It assumes that the desired mesh is such that each point of it has one and only one projection according to a *predefined view.* In other words, it assumes that a perspective, orthographic, cylindrical or spherical projection exists such that the desired mesh never hides parts of itself if seen from that projection.

In order to create the mesh you need:

1) *An ustructured or structured point cloud*: drag from the Project window the point cloud you want to mesh.
2) *A view*: you need to indicate also which *view* should be used to build the mesh. The current view can be used, otherwise you can use any camera from your project (see below for some examples).

Create a mesh from an orthographic view: a façade	*Create a mesh from a predefined cylindrical view: a tunnel*





When the cloud and the viewpoint are selected, press **Ok** and wait for the end of the meshing. You can also cancel the meshing while it proceeds. At the end of meshing, a dialog shows you the new mesh's name and properties.

This algorithm works by projecting all the points of the cloud on a 2D surface: the near plane of the selected projection. Then, these points in 2D are meshed with a standard 2D meshing algorithm: Delaunay. The resulting mesh is then projected back into the 3D space.
This meshing technique is *interpolative*: each point of the input cloud is considered as vertex of the output mesh. Reconstructor®'s Multiresolution and Topographic meshes are, instead, *approximative* meshing techniques: some points can also not be mesh vertices if they don't significantly modify the surface's shape. For this reason, this interpolative technique is generally slower than approximative ones, and produces heavier meshes. However, it always produces watertight meshes.
Heavy meshes or meshes with wrong shapes can be simplified and edited in the mesh editor.

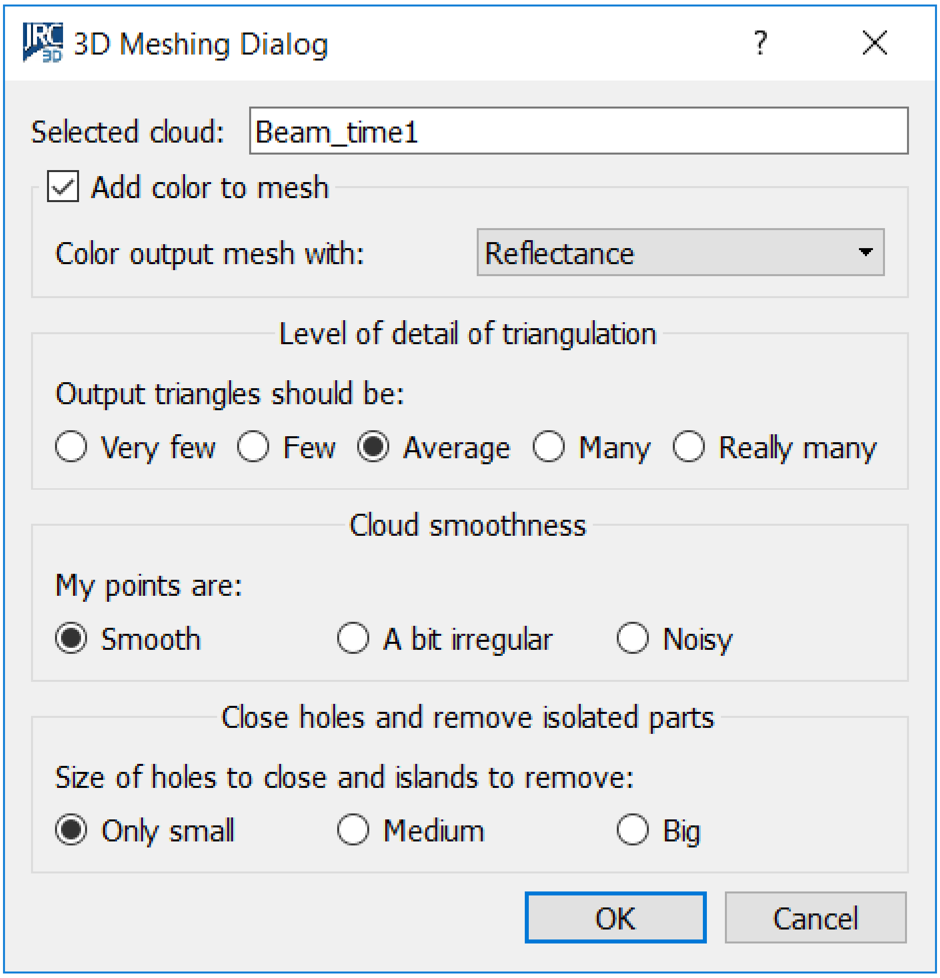
See also other Meshing techniques.

3D Meshing (Poisson reconstruction)

This tool allows you to reconstruct a triangle mesh from any point cloud. Unlike uniform or multiresolution meshing, this technique does not need the input cloud to be structured. Also, unlike meshing from predefined view and topographic meshing, this technique is not view-dependent. It reconstructs a connected, watertight surface taking as constraints the points' positions and orientations (normals).
It's suggested to reconstruct a convex (closed) surface.

If you need to work with a set of point clouds (structured or not), it's better to lump them together in an unstructured point cloud using the Level 3D density of clouds command.

In the 3D Meshing Dialog you can manage some parameters' presets to obtain a best fitted mesh.



Adding color
In the first parameter box of the dialog, you can select by the flag whether the output mesh should be colored or not, and which of the input cloud's color layers should be used.

Resulting mesh resolution
While meshing from predefined view is *interpolative*, this reconstruction technique is *approximative:* it creates a triangulation that approximates the surface described by the points. In the second parameter box, you can specify how many details you want to preserve in the output mesh. The higher the resolution (from *Low* to *High*), the finer the details that will be reconstructed, and also the heavier the mesh file and rendering time.

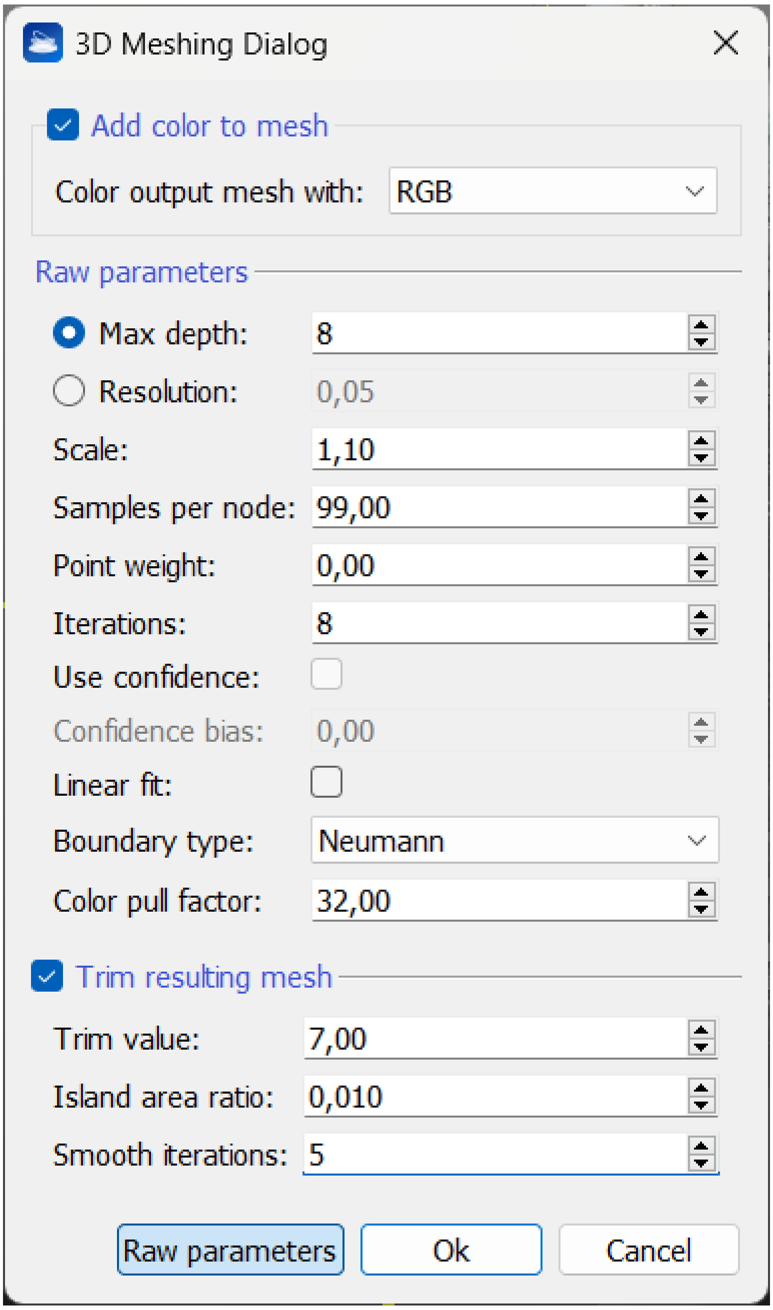
Input cloud noise level
In the third parameter box, you can take into account the cloud smoothness, according to the point cloud noise.

If your cloud represents clean construction data, then select *Low*. If, however, your cloud contains noisy territorial data, then select *High*.

Size of holes to close and islands to remove
In the last parameter box, you can specify the size of the holes that are going to be closed in the mesh, and the size of the “islands” (small disconnected components that may have been created) to be removed (or not). The size is computed as ratio between the area of the hole/island and the area of the whole mesh.

Raw parameters

By clicking the *Raw parameters* button, a new dialog expliciting all the parameters will appear.



- *Max depth*: it is the maximum depth of the tree that will be used for surface reconstruction, defining the resolution of the final mesh. Running at depth d corresponds to solving on a grid whose resolution is no larger than 2^d x 2^d x ... Note that since the reconstructor adapts the octree to the sampling density, the specified reconstruction depth is only an upper bound.

  Generally, the bigger is the value, the better is the reconstruction - preserving more details - but more time is necessary. For almost all uses keep the value between 8 and 12, and not to increase it more than 20 because it is usefulness and no details are added.
- *Resolution*: as an alternative to the *Max depth* parameter, the resolution can be directly set changing this value defining the density of the initial point cloud.
- *Scale*: it specifies the ratio between the diameter of the cube used for reconstruction and the diameter of the samples' bounding cube. The default value is 1.1.
- *Samples per node*: it defines how to smooth the final mesh. SPN is the minimum number of sample points that should fall within an octree node as the octree construction is adapted to sampling density.

For noise-free samples, small values in the range[1.0-1.5] can be used. For more noisy samples, larger values in the range [15.0-20.0] may be needed to provide a smoother, noise-reduced reconstruction.

- *Point weight*: it defines the capacity of the algorithm to preserve the details.

  If you have small disalignments or "double surfaces" in the point clouds, it's suggested to set small values, from 0 until 4. Otherwise set 4 or larger values.
- *Iterations*: it specifies the number of Gauss-Seidel relaxations to be performed at each level of the hiearchy. The default value for this parameter is 8.
- *Use confidence*: flagging this parameter, the quality of the points (defined by the confidence value, if present in the initial point cloud) will be used to obtain a better reconstruction. The normals are here normalized using the confidence values.
  Confidence bias: it specifies the exponent to be applied to a point's confidence to bias the resolution at which the sample contributes to the linear system. Points with lower confidence are biased to contribute at coarser resolutions. The default value for this parameter is 0.
- *Linear fit*: enabling this flag has the reconstructor use linear interpolation to estimate the positions of iso-vertices.
- *Boundary type*: it specifies the boundary type for the finite elements. choosing between Free boundary constraints, Dirichlet boundary constraints and Neumann boundary constraints. The default value for this parameter is Neumann.
- *Color pull factor*: if the input points have additional data as color, that is to be sampled at the output vertices, this value specifies the relative importance of finer data over lower data in performing the extrapolation. The default value for this parameter is 32.

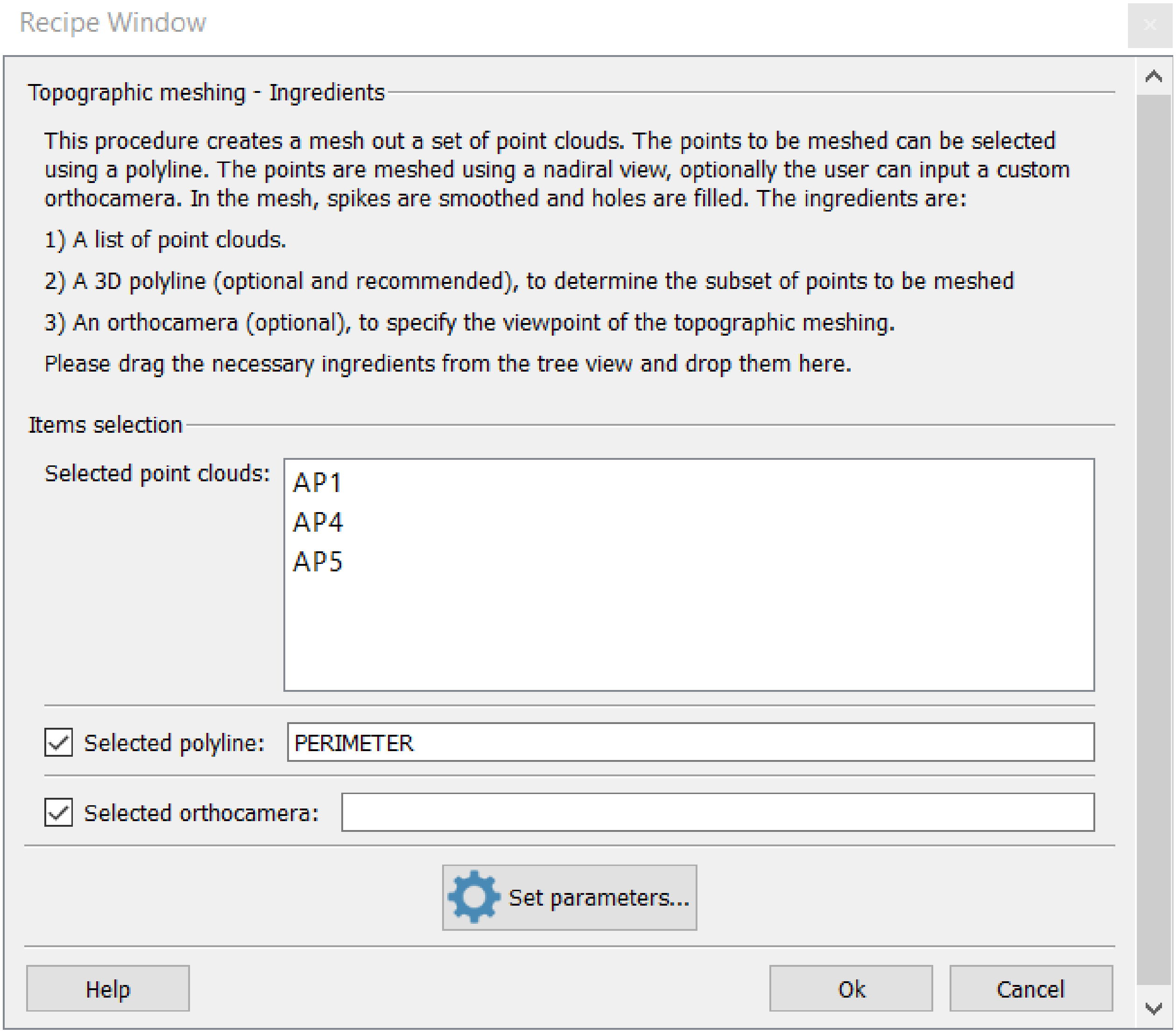
By flagging ***Trim resulting mesh*** it is possible to reduce the output mesh extents so as to match as best the input cloud extents. The triangles with vertices having the lowest density values, that are the farthest from the input cloud, will be detected and removed.

- *Trim value*: it specifies the value for mesh trimming. The subset of the mesh with density value less than the trim value is discarded.
- *Island area ratio*: it specifies the area ratio that defines a disconnected component as an "island". Connected components whose area, relative to the total area of the mesh, are smaller than this value will be merged into the output surface to close small holes, and will be discarded from the output surface to remove small disconnected components. The default value 0.001.
- *Smooth iterations*: it values the number of umbrella smoothing operations to perform on the density cloud before trimming. The default value is 5.

NOTE: the algorithm is based on the *Poisson Surface Reconstruction code* developed by Misha Kazhdan et al.

See also other Meshing techniques.

Topographic meshing



This function implements topographic meshing of a set of point clouds. This procedure takes a set of point clouds and creates a DTM regularly sampling the clouds. The resulting mesh is watertight if the default parameters are used, and it is colored according to the *altitudes*: points at minimum height are in red and points at maximum height are in violet, passing through all hues.

The user can also input a polyline to determine the points to be meshed (seen from a nadiral view or from a user-specified orthocamera). It is strongly recommended to input a polyline in order to help Reconstructor to concentrate only on the useful points and to speed up the process.

If the user doesn't specify any orthocamera, one is created so that it is nadiral (oriented towards -Z axis in the current UCS) and it contains the polyline. Once the user inputs the clouds and – optionally – the polyline, meshing starts by pressing Ok.

The procedure is composed of the following stages:

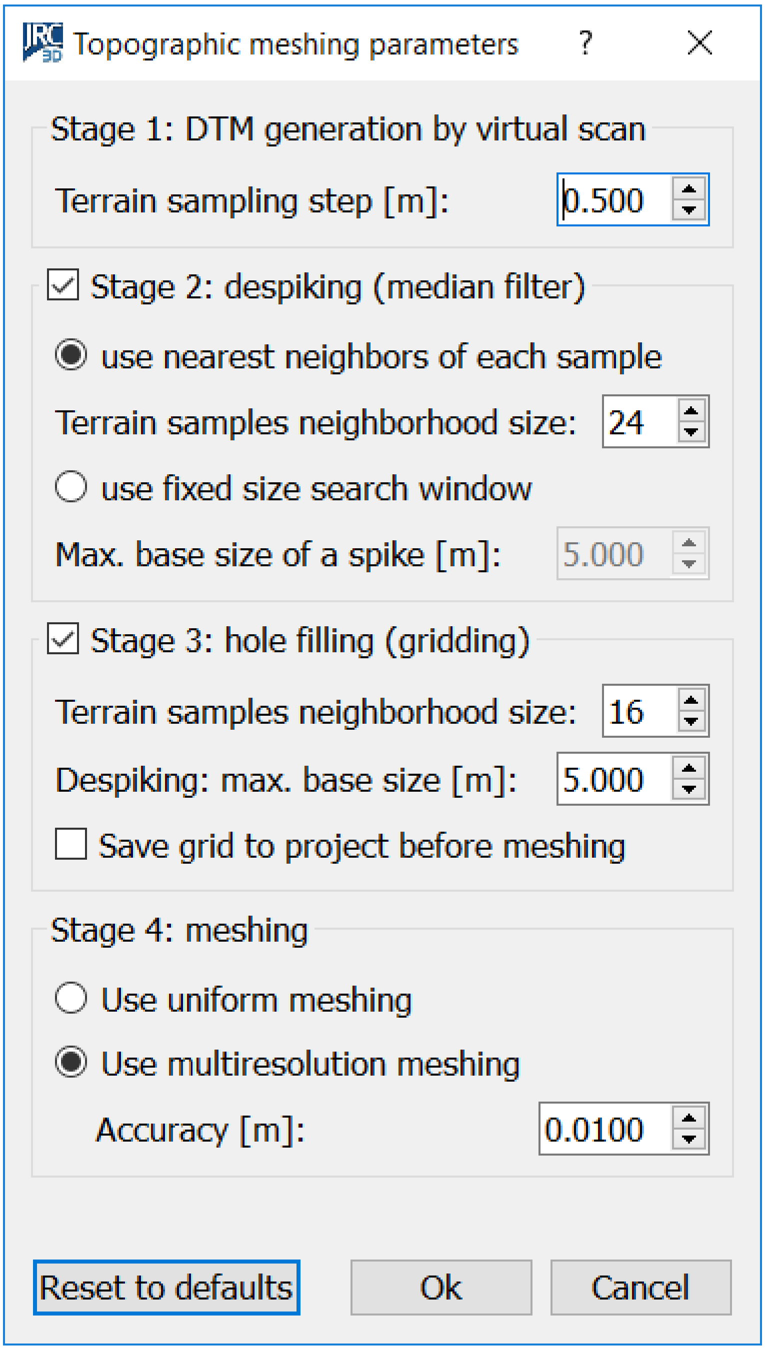
- Virtual scan is performed from the (nadiral) orthocamera, to uniformly sample the point clouds.
- Then, spikes in the resulting samples are smoothed using a median filter based either on a neighborhood of given size, or on a window of fixed size.
- Successively, a polynomial interpolation (ordinary gridding) is used to fill holes. This eliminates holes in the samples and guarantees the result mesh to be watertight.
- Then, a part of the obtained grid point cloud is selected using the polyline and the orthocamera.

The selection with polyline allows the user to specify the region that he/she wants to be meshed.

- Subsequently, a median is applied to the resulting grid, to smooth it out.
- Finally, the resulting mesh is obtained by multiresolution meshing, filling all holes.

The mesh is saved in the project, under the name *Topographic mesh of <n> clouds*.
This procedure is relatively fast and useful especially for terrains. It outputs a mesh with nice properties: watertight, light and smooth (without spikes). The procedure can be customized by clicking the *set parameters* button.

Topographic meshing parameters



In this dialog you can set the parameters of the topographic meshing procedure.

This dialog shows from top to bottom the four stages of the topographic meshing procedure. You can customize the parameters of each stage, and even decide to skip stages 2 and 3 by unchecking the respective boxes.

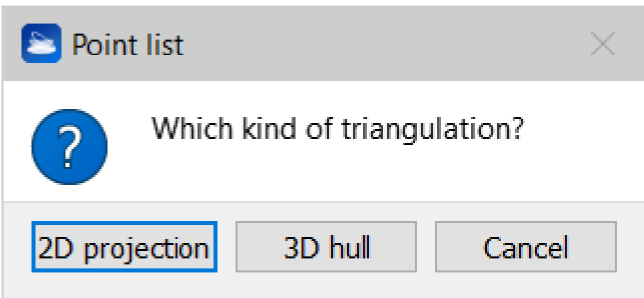
- **Stage 1: DTM generation**. The *terrain sampling step* is the fundamental parameter of this procedure. It is important to pay attention that this step is meaningful in relation to the dimensions of the desired model. If I want a mesh of a terrain of 20 m^2, I shouldn't use a sampling step of 0,5 m, otherwise I'll get too few samples. A small step will produce more accurate and heavier meshes. A big step will give less accuracy and also less memory occupied by the final mesh.
- **Stage 2: despiking**. Despiking is very useful to delete noisy data (e.g. vegetation) and therefore to have more precise calculations of volumes and isolines on the resulting mesh. Despiking is done with a median filter running on the grid point cloud obtained in the former step. As the dialog shows, the median filter can run on the *N nearest neighbors* of a sample, or on a window of samples of fixed size, regardless of how many samples are actually in the window.
- **Stage 3: hole filling**. This is performed by taking, for each missing sample in the grid point cloud, the <*neighborhood size*> samples closest. Polynomial interpolation is used to decide the height position of the missing sample. After the hole filling, another median filter is run on a square of samples of fixed size. The user can save the grid point cloud obtained till here by checking *Save grid to project before meshing*. This can be useful for example to mesh the grid using a color not related to the altitudes.
- **Stage 4: meshing**. In the end, the grid point cloud is meshed using one of the two techniques described here. Since at this stage the grid point cloud should be smooth, regular and without holes, there is no reason to change these parameters except for very particular cases.

Use the *Reset to defaults* button on bottom left if you are not sure of what is happening. However, topographic meshing is so fast that it is worth to try many combinations of parameters and compare the results.

See also other Meshing techniques.

Mesh point list

This options creates a triangle mesh using as vertices the points listed in a point list.

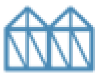
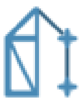


Reconstructor asks you whether you want to mesh the points projecting them onto the current view and using a Delaunay 2D meshing algorithm, or whether you want to create a “3D hull”: a convex 3D polygon containing all the points.

Mesh manipulation

In this section you can first learn how to edit triangle meshes to select and to filter them and so to obtain a better result from your data.

Reconstructor® offers several functions for editing meshes:

	**Mesh Editor**	An editing environment to perform advanced operations such as hole-filling, borders detection, editing triangles and vertexes, smoothing surfaces, decimating, crests and toes extraction.
	**Mesh selection from current view point**	To cut a portion of a mesh using 2D video selection tools on the current view.
	**Mesh selection with 3D polyline**	To cut a portion of a mesh using an input mesh, a 3D polyline and a viewpoint
	***Merge meshes into one***	This dialog allows to lump together in a single mesh an arbitrary set of triangle meshes
	**Convert mesh to point cloud**	To create an unstructured point cloud from the vertexes of the mesh using the color attribute of the mesh
	**Get mesh borders as polyline**	To create a new polyline containing the mesh's borders and add it to the project

From the item's context menu other commands are available:

- **Compute mesh normals**

To compute or update the triangles' normals for the mesh.

- **Invert winding**

This command inverts the ordering of the vertexes for each triangle, so the surface is flipped to the opposite side and also the normals are inverted.

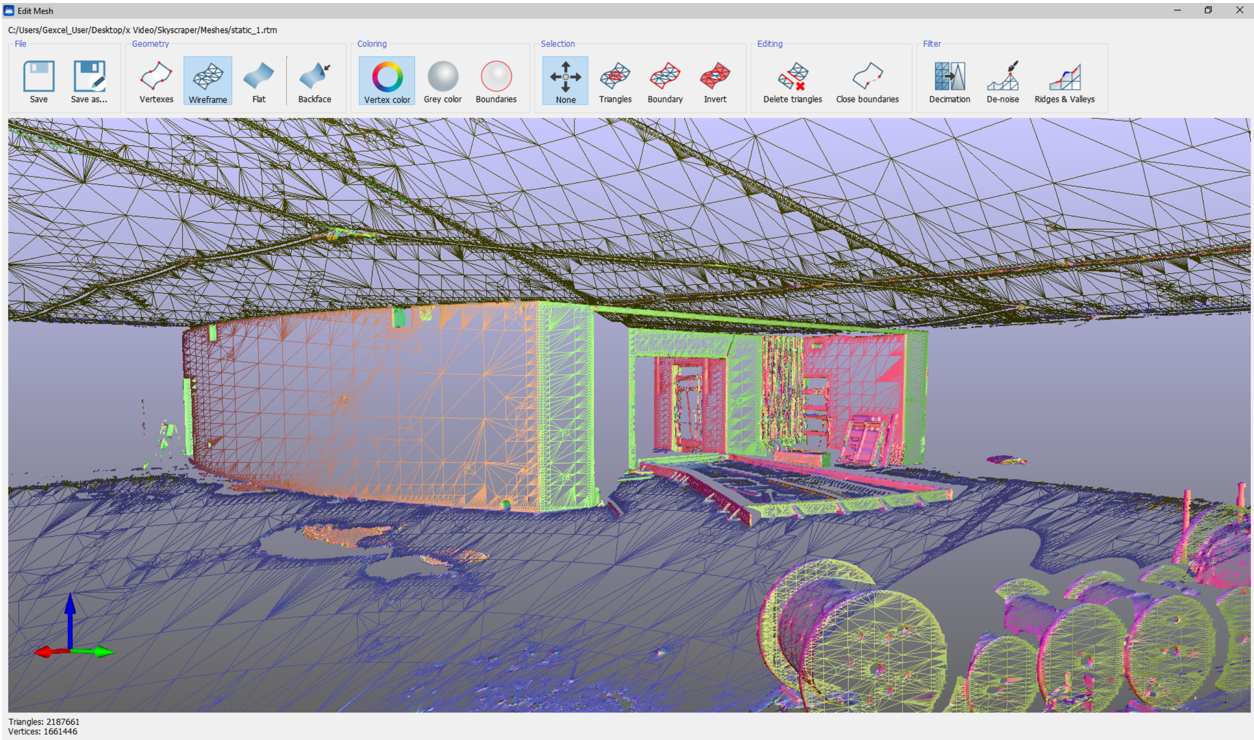
- **Compute area**

This command returns the mesh area as sum of the areas of all the mesh's triangles.

- **Compute volume from Z=0 plane**

This command returns the volume resulting from integrating the mesh on the XY plane of the current UCS. Mesh triangles below the XY plane will result in zero volume.

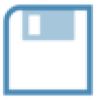
Mesh Editor



This dialog is an environment designed for advanced mesh editing operations. It works on one mesh at a time. It is accessible from *Data management→Meshing→Mesh editor*, from the corresponding button in the top toolbar, or from the context menu of any mesh in the project window.

From top to bottom, the dialog displays the mesh's full path, then a toolbar with groups of buttons, then a navigable 3D view to show the mesh, and lastly information on triangles and vertices count.

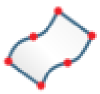
File



This button group provides the *Save* and *Save as* options. These are needed because the edits performed in this dialog are not automatically transferred to the mesh in Reconstructor®'s project.
In fact, if you close the dialog you are asked if you want to save or discard the changes.

Geometry

This button group allows you to switch between the three visualization possibilities of a mesh:



only vertices,

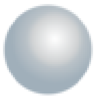
only triangle edges (wireframe),

or the triangle's faces (flat).

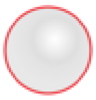
The button *backface* allows you to show or hide the triangles' backface.

Coloring

and



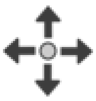
The first two buttons allow you to change how to color the mesh: using the colors associated to the vertices, or using a gray color that changes only according to the triangles' inclination with respect to the light source.



The button boundaries highlights in bright red all the boundary edges of the mesh, and it is particularly useful to quickly spot where the mesh has holes to fill or imperfect*boundaries* that should be corrected.

Selection

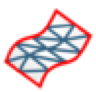
Here you can use several selection modes.



When the button *None* is checked, you can navigate the mesh.

By clicking *Triangles*, you start selecting the mesh triangles by drawing a rectangle on the viewport. Selected triangles are highlighted in yellow.

The button *Invert* enables you to invert the current triangles selection.

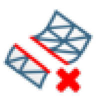


The button *Boundary* turns on another selection mode. In this mode you can select one of the circular boundaries that the mesh has. Boundaries are polygons formed by those mesh edges who belong to one and only one triangle. A closed mesh has no boundaries. Boundaries selected in this mode may delimit a hole that you want to close. After selecting the hole's boundaries, you can click on *Close boundaries* in the Editing button group.

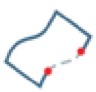
By pressing the button *Close* all the triangles will be deselected.

When selecting and managing triangles the *Flat* geometry must be selected in order to visualize the triangles' surfaces colored in yellow after the selection process.

Editing



Once you have selected some triangles with the command *Triangles*, you can delete the selected triangles with *Delete triangles*.

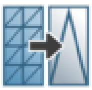


By using the *Close boundaries* button you can close a mesh hole whose boundaries have been selected with *Boundary* command.

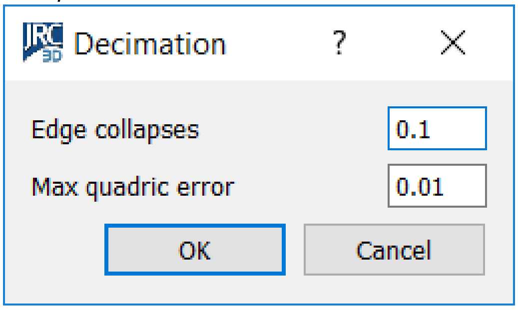
This function works best with small and with close-to-planar holes.

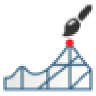
Filter

This button group contains four functions:

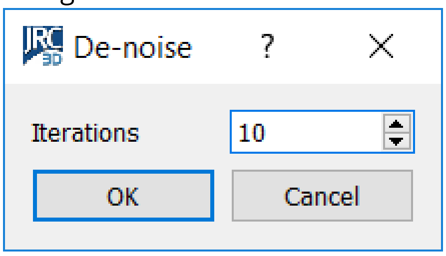


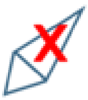
The *Decimation* tool allows you to simplify your mesh substituting small triangles with bigger ones if the mean-square error introduced in this way is smaller than a user-customizable threshold.. Simplification is based on conveniently *collapsing edges* among triangles.
For each edge shared by two triangles, Reconstructor® assesses whether collapsing it or not. Collapsing an edge means assigning the edge's two vertices to coincide in the edge's middle point, and adjusting all neighboring triangles consequently. If the quadratic error between the surface before and the surface after the collapsing is below the parameter *Max quadric error* of the dialog, then the edge is collapsed.
The algorithm tries to collapse a percentage of edges as specified by the parameter *Edge collapses*, where 0.1 stands for 10%.





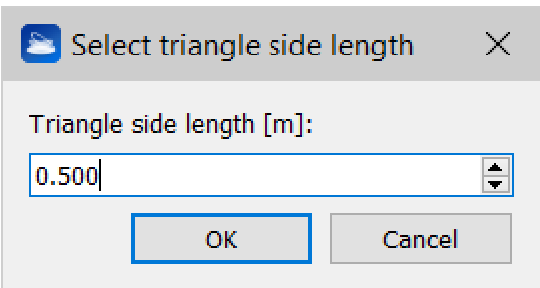
This *De-noise* algorithm allows you to *de-noise* your mesh, meaning to iteratively eliminate spikes from your mesh, without changing the surface's volume.
A smoothing filter is run on the mesh for as many times as the parameter *Iterations* says. This filter has *anisotropic* properties: it eliminates the spikes without changing the surface's volume. Along the iterations, it distributes the volume from the spike to the neighboring triangles.





The *Triangles remover* tool select the triangles that have at least one side greater than the entered threshold value *Triangle side length* in meters.

It can be useful to remove stretched triangles in correspondence of areas of discontinuity or large triangles at low density on the boundaries.

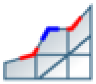


The triangles corresponding to this filter will be colored in yellow in the 3D view. You can remove them by using the command *Delete triangles*.

The *Trim mesh* tool select the triangles along the boundaries of the mesh. These triangle have at least one side not connected with another triangle.

The triangles corresponding to this filter will be colored in yellow in the 3D view. You can remove them by using the command *Delete triangles*.

If the MINING add-on is activated, also the *Crest and toe* command will be included in this section. You can access it through the dedicated command in the MINING toolbox bar.



Crest and toe is a semi-automatic technique to detect *crests* (prominent mesh edges) and *toes* (reentrant mesh edges). When you open this dialog, the crests and toes are calculated and drawn in the mesh editor. Crests are drawn as blue polylines, Toes are in red. Crests and toes can be then saved as polylines.

The idea is that tool shows you an initial extraction of crests and toes, successively you refine the parameters of extraction and see the effects of your changes in real time. You can refine the parameters until you are not satisfied, and then save the crests and toes as polylines in the project. The parameters of crests&toes extraction you can modify regard the *curvature*, the *horizontality*, the *length* of the edges and the *gaps* between them. While you modify the parameters and recompute the crests&toes, you can choose to turn on or off the visualization of the crests, the toes and the mesh, via the three check-boxes in the *Draw* groupbox at the bottom of the dialog.

Curvature filter

The topmost slider of the dialog allows you to modify a threshold on the *curvature* of the crests and toes. Crests and toes are edges shared by your mesh's triangles. Each edge has a curvature associated, depending on the angle between the two associated triangles. A threshold on the curvature filters out the smoother edges and leaves only the steeper ones, which normally are the most important. Scroll the slider to see the effect of the threshold in real time.

Orientation filter

Below the curvature filter, you can also activate a *orientation* filter. This filter keeps only the edges whose angle with respect to the horizontal plane is close to the angle you indicate with the slider. In this way, you can choose to keep only the horizontal edges, or the vertical ones, or the ones with a given inclination.

Close gaps in edges

Sometimes crests and toes come in many segments with small gaps among them. In the groupbox *Close gaps in edges* you find a tool to close those gaps and get the crests and toes as continuous polylines. You can set the maximum gap distance to close, and press *Apply*. Reconstructor will start closing the gaps from the smallest to the biggest, without creating loops or bifurcations.

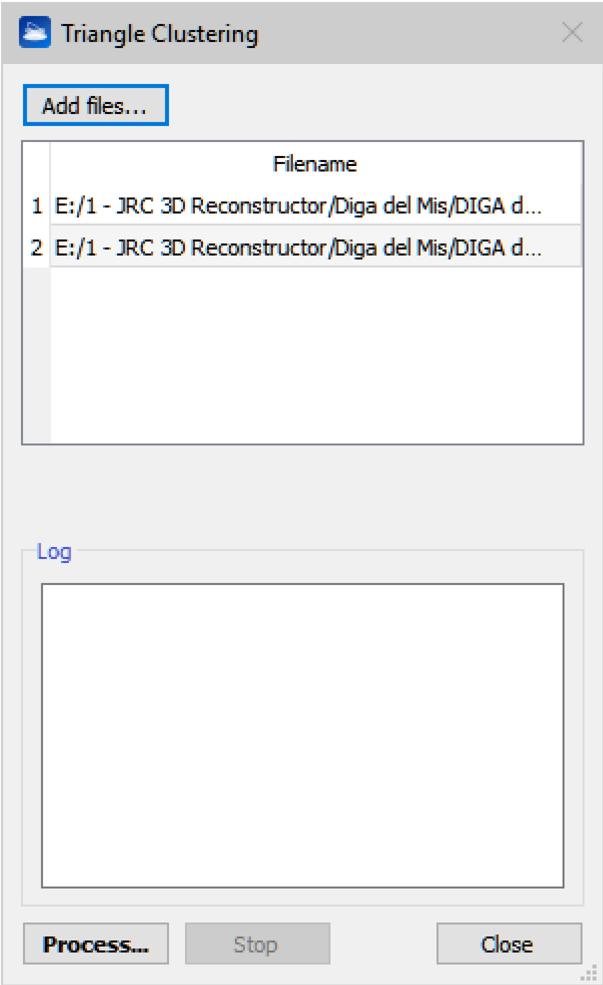
Delete short edges

In the *Delete short edges* a tool comes to filter out polylines whose total length is less that the specified parameter. Normally you want to discard short edges because they are not the main ones. The main ones should instead be found by closing the gaps among them.

General procedure

As general procedure, we advice to use the described filters in the order given above, and to close and open the dialog if the result is not satisfactory. Once the extracted crests and toes are good enough to describe the important edges of your model, press *Save as polylines* at the bottom of the dialog. New polylines with the crests and toes will be added to your project.

Make single mesh



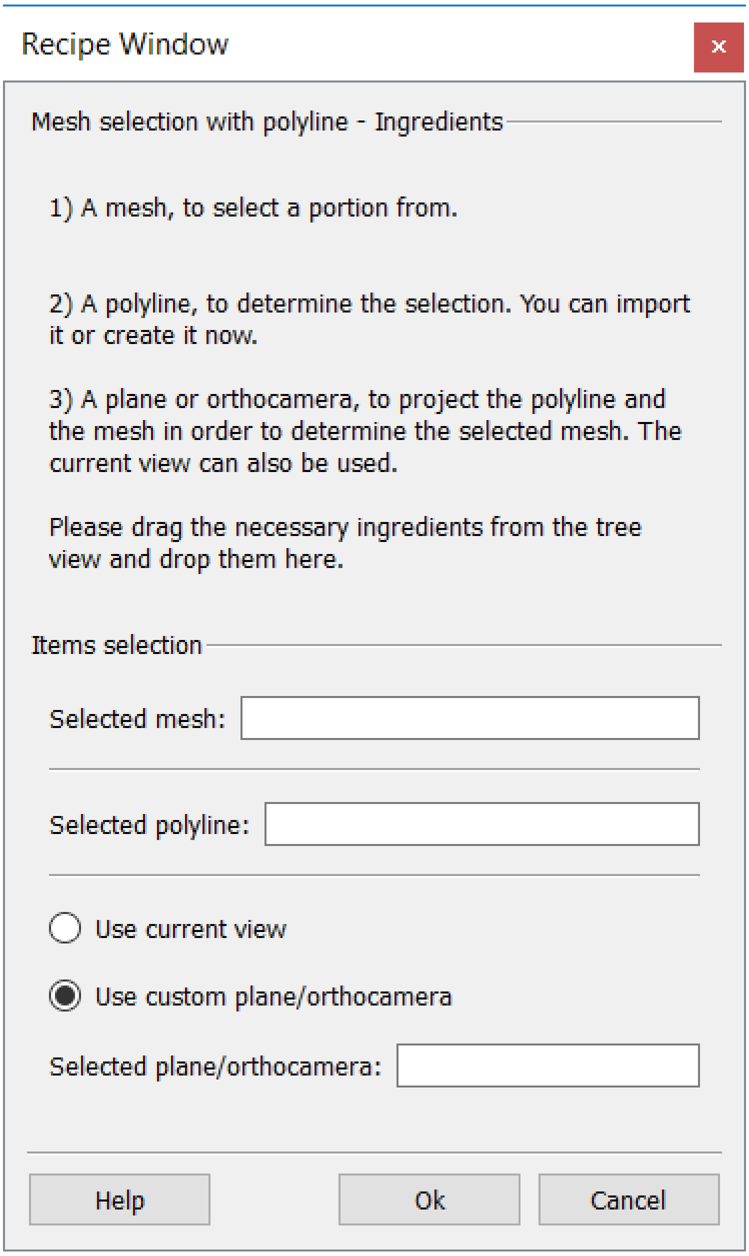
This dialog allows to lump together in a single mesh an arbitrary set of triangle meshes.

To activate this dialog, you can select a set of triangle meshes from the project window and select *Meshing ->Make single mesh* from the contextual menu or from the *Tools* menu.

To add more meshes to cluster together, click *Add files* on top left to open point mesh files (.rtm).
By clicking *Process* on bottom left, you start the clustering.
A new mesh is created that simply contains all the triangles of the input meshes.

See also triangle meshes.

Mesh selection with 3D polyline



The 3D mesh selection works with an input mesh, a 3D polyline and a viewpoint that can be defined by an orthocamera, by a plane or by the current view.

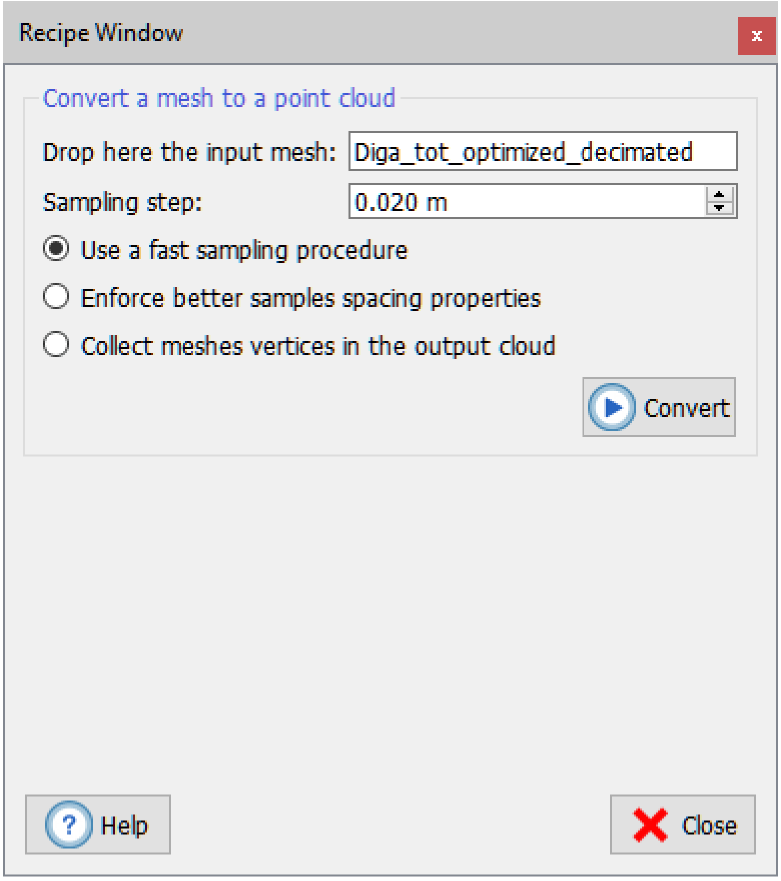
The polyline, seen by the input viewpoint, determines a frustum that intercepts the mesh defining a portion of it. This portion is acquired by doing Virtual scan, and uniformly meshing the resulting grid point cloud. The virtual scan is always done with an orthocamera that Reconstructor® creates internally.

When selecting the polyline and the viewpoint, the user doesn't need to check whether the polyline is contained in the orthocamera's frustum, or how the polyline is positioned with respect to the plane. Reconstructor, in fact, creates an orthocamera whose frustum is automatically enlarged to include all the polyline. This automatic expansion, however, does not take into account the mesh (this is done to fully exploit the orthocamera's resolution on the selection region).

The resolution of the internal orthocamera is calculated proportionally to the input mesh's vertices amount, so that the scanning is precise enough to preserve the input mesh's features.

If a plane is given as input, Reconstructor® creates an orthocamera that points in the plane +Z direction. If the polyline and the mesh are in the -Z semispace, they will be included anyway in the selection because the orthocamera will have a negative near plane. However, the direction of the plane (or of the orthocamera) influences the winding direction of the mesh that results from the selection.

Convert mesh to point cloud



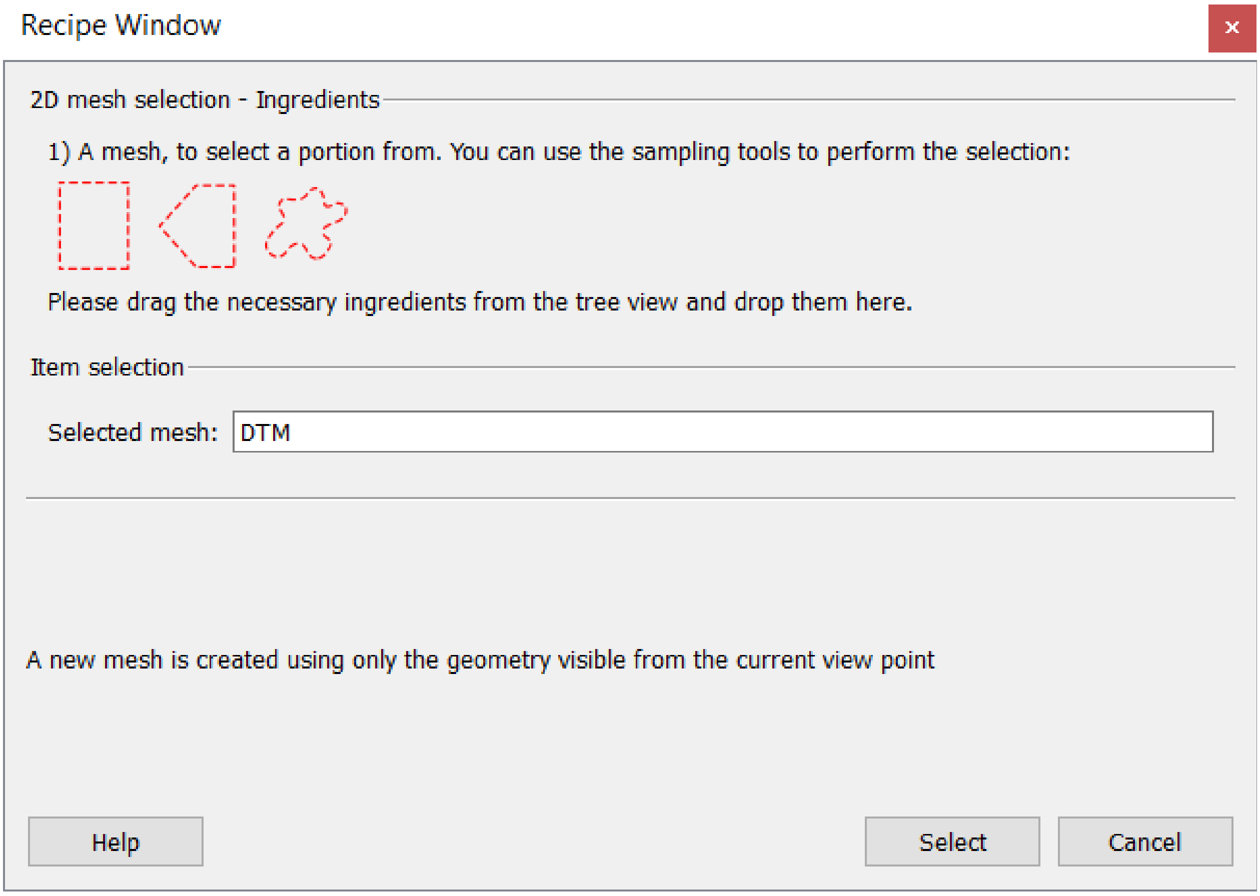
This recipe window allows the user to transform a mesh into an unstructured cloud of points.
Three methods are available:

- *Use fast sampling procedure* converts the mesh picking a point about every sampling step.
- *Enforce better sample spacing properties* converts in a better way the mesh, respecting more faithfully the sampling step; it takes longer than the previous method and requires more amount of RAM. Not recommended for large meshes or for very high sampling.
- *Collect meshes vertices in the output cloud* simply transforms the vertices of the mesh into points of the resulting cloud

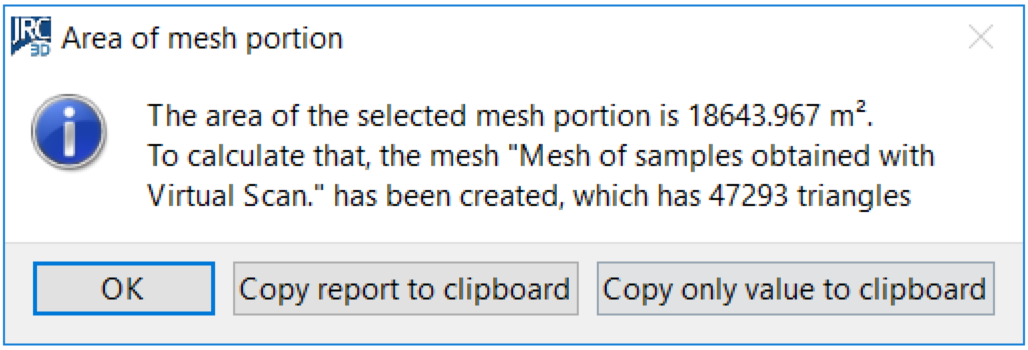
Get mesh border as polyline

This tool is used to create a new polyline containing the mesh's borders and add it to the project.

Mesh selection from current view point



This function allows to cut a portion of a mesh using 2D video selection tools on the current view. It is accessible through *Tools-> Meshing->Mesh selection from current view point*. When you activate this function, the window above appears in the recipy window, and Reconstructor®'s 3D window goes in *Selection* mode.
Drag a mesh from the project window and drop it on the window shown above. The mesh must be loaded for the procedure to work. Use one of the three video selection tools to select the desired portion of mesh, and press Select. The dialog shown below appears:

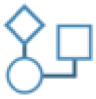


The dialog above shows the name of the mesh created from the selection, and information about area and triangles count. After closing the dialog, you may want to go back to *Navigation* mode by, for example, pressing the space bar.

Geometric Shapes

Points, Segments, Plans, Circles, Cones, Cylinders, Spheres, Polylines: each type of geometric shape has a special recipe window for its creation, fitting and editing.

Manage Points
To edit and use points lists.



Basic Shapes
To create and edit segments, polylines and planes.

Second Order Shapes
To create and edit second order shapes like circles, spheres, cylinders and cones.

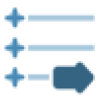
Shapes Interactions
The geometric shapes can be interact between them to extract intersections and distances.

Manage points

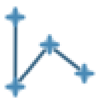
- All the points picked or imported in Reconstructor® can be edited and used to extract shapes.

The below tools can be found into *Data management* > *Geometric Shapes* Top menu and in the *Manage points* menu in the Top toolbar.

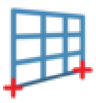
Edit point list



Export point list



Polyline from point list



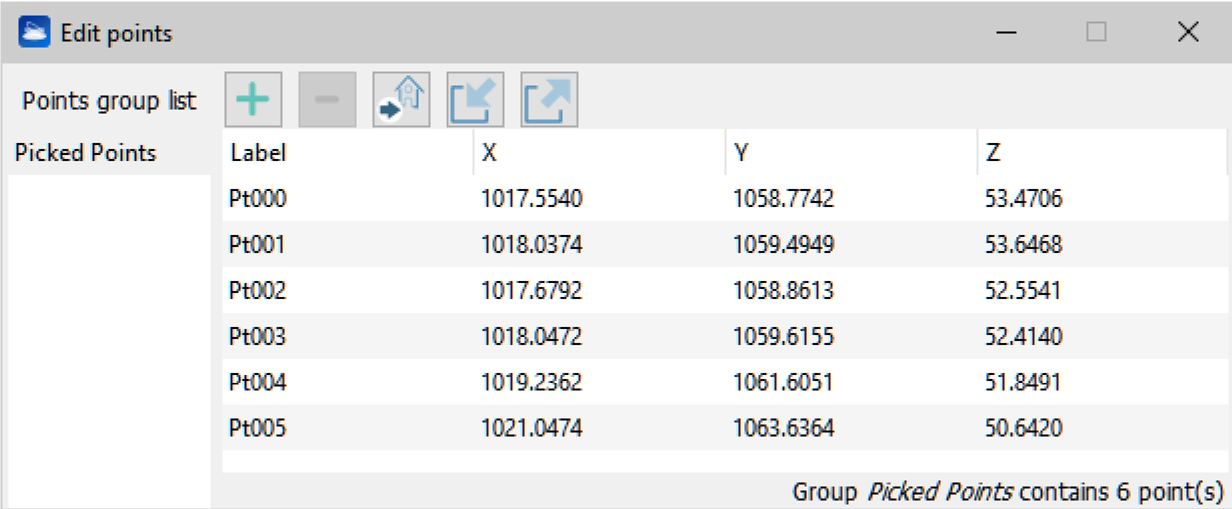
Fit shape to point list

The point list context menu

In addition of the previous ones, other tools are achievable in the point list's context menu:

- **Mesh point list**
- **Flythrough from point list**
- **UPC from point list**: the selected list of points is saved as an Unstructured Point Cloud and added to the current project, so it can be exported to other formats.

Edit point list



The point list window works as a collector of selected points, which can then be used for various purposes: defining a polyline, a mesh, a video trajectory, creating a plane, calculating distances, performing point-based georeferentiation, etc.

While navigating the 3D scene, with ***Alt+left mouse button double click***you can select points on your 3D models. The selected points appear annotated in the 3D scene and listed in a group called *Picked points*. You can create new groups and move on them some points to create other point lists.

The toolbar of the Edit points window

Once you have collected or imported the points, you can perform several operations on them. These are accessible through the point list window toolbar. Below, the toolbar buttons are described from left to right:

- **Add a point**: adds an empty row in the list, to input manually 3D coordinates.
- **Delete selected points**: removes the currently selected row in the list. With [Ctrl] or [Shift] keys multiple rows can be selected, and deleted with the DEL key.
- **Go To**: move the view next to the point
- **Import a list of points**: loads a list of points by parsing them from a text file via the parse point list tool.
- **Export points**: saves the point list to a text file, writing one row per point in the format label, x, y, z. See Export point list for file formats.

Export point list

This function exports a list of points (in *.txt, *.csv, *.tsv formats) including labels and coordinates (in the current UCS).
It is available in the context menu, in the *File* > *Export* menu, in *Meshes* > *Manage points* and in the Edit Point List command.

Only few points can be exported, according to the criteria:

- If a group of points is selected in the Project Window, before to select the *Export point list* tool, only the points in this group will be exported
- If the "Picked Points" group is selected in the Project Window, before to select the *Export point list* tool, all the points (in this group) will be exported

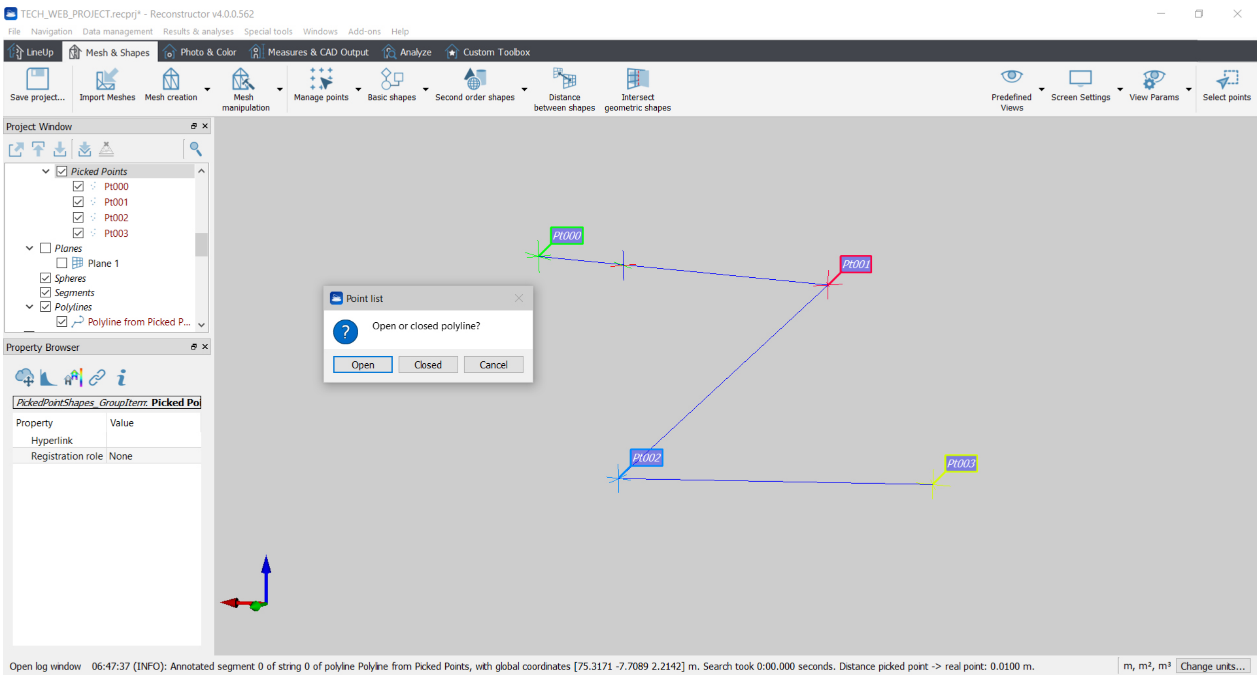
- If few points are selected (by pressing *Ctrl* button) in the Project Window, before to select the *Export point list* tool, only these selected points will be exported.

Three formats are available:

- Comma Separated Value (*.csv)
  Comma "," is used as separator.
  ```
  Pt000,-13.9153,-15.7213,11.9317
  Pt001,-36.6361,15.7372,6.3014
  Pt002,-15.1782,6.6534,8.5348
  ```
- Tab Separated Value (*.tsv)
  Tab "*Space*" is used as separator.
  ```
  Pt000   -13.9153   -15.7213   11.9317
  Pt001   -36.6361   15.7372    6.3014
  Pt002   -15.1782   6.6534     8.5348
  ```
- Text (*.txt)
  Tab "*Space*" is used as separator.
  ```
  Pt000   -13.9153   -15.7213   11.9317
  Pt001   -36.6361   15.7372    6.3014
  Pt002   -15.1782   6.6534     8.5348
  ```

Polyline from point list

This option will store all the points in a polygon and save it as a polyline in the project. Reconstructor® asks you whether you want to create an open or close polyline.



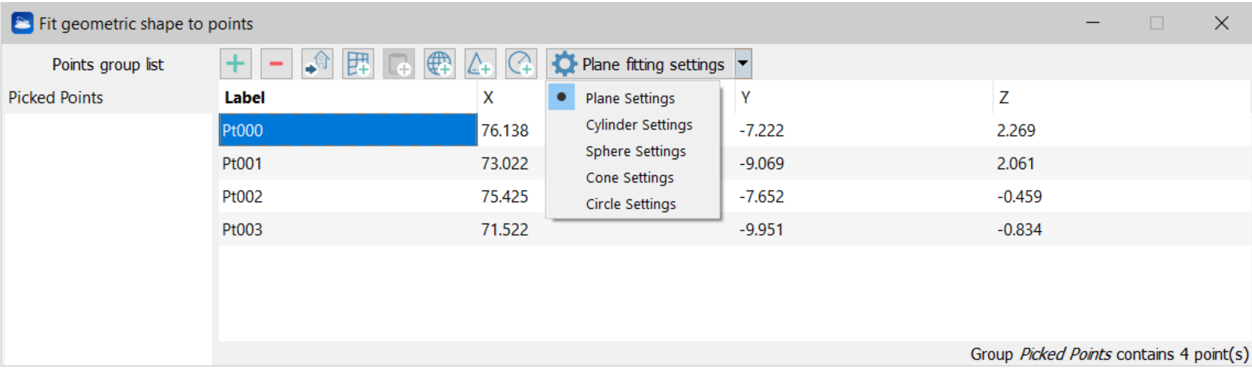
The polyline will be created according to the criteria:

- If the "Picked Points" group is selected in the Project Window, before to select the *Polyline from point list* tool, all the points (in this group) will be used
- If few points are selected (by pressing *Ctrl* button) in the Project Window, before to select the *Polyline from point list* tool, only these selected points will be used

- The selected points will be joined in sequence as they appear in the project window; the first and the last points will be joined when a polygon is closed.

Fit shape to point list

If at least 3 points are defined, a fitting shape can be computed and added to the project.



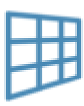
According to the number of the selected points a plane, a cylinder, a sphere, a cone or a circle can be fitted.

For each shape, a default set of *fitting parameters* can be adopted or modified to interpolate the points.

Among the selected points, the RANSAC algorithm randomly selects X points and tries to fit the geometric shape on them.
The algorithm then iterates this procedure N times keeping the best result that it found among all attempts. The quality of the fitting is given by how many points (out of the selected ones) are sufficiently close - and therefore belonging - to the geometric shape (*inliers*).
The maximum value of X is given by the *Maximum points used for fitting [#]* parameter.
The *RANSAC minimum iterations [#]* parameter defines the minimum number of attempts N that the algorithm must make before considering the geometric shape correctly fitted.
Inliers maximum distance [m] defines the maximum distance within which a point is considered to belong to the geometric shape.

A special button allows you to reset to default parameters at any time.

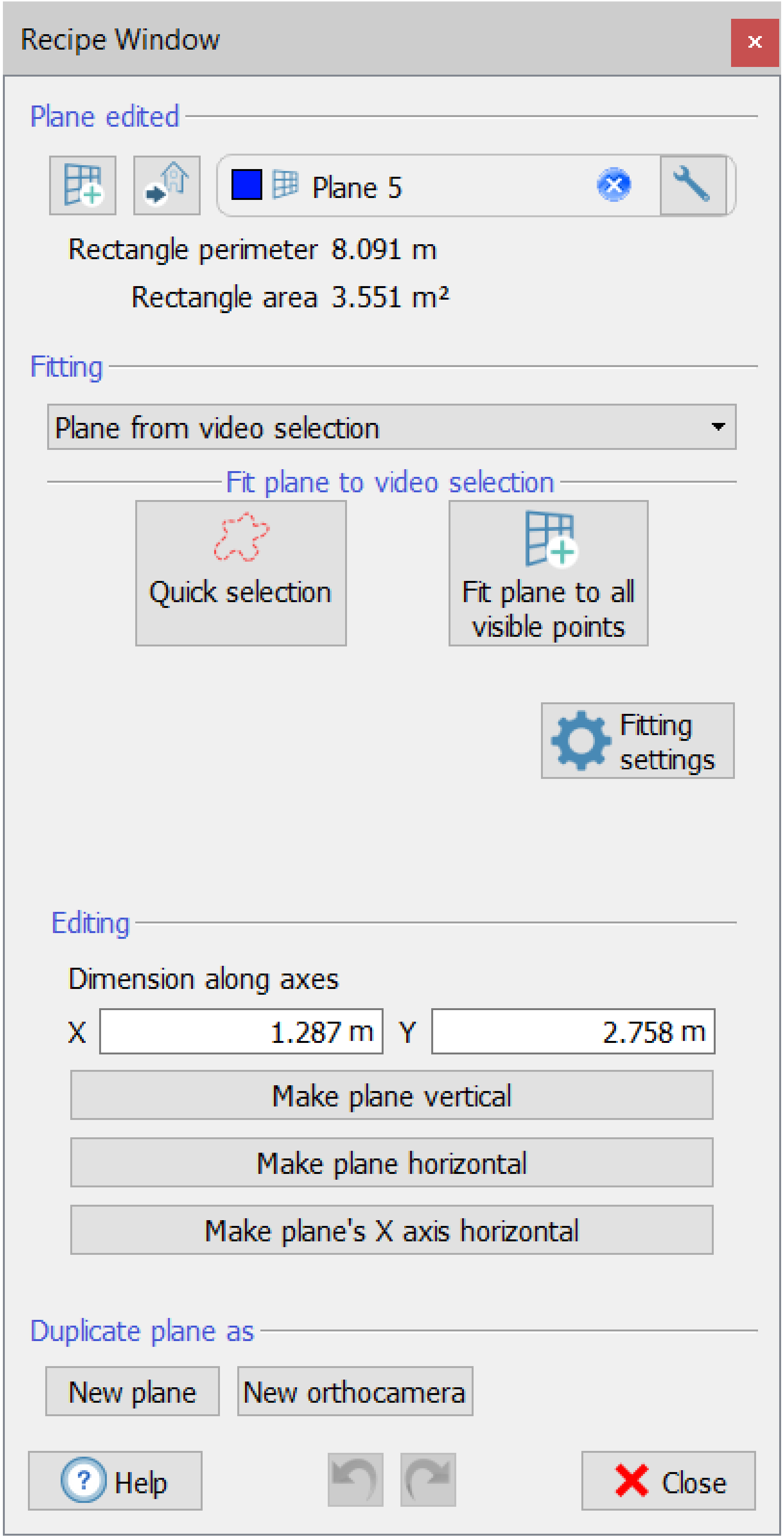
Basic shapes

 **Create/edit plane**

 **Create/edit segment**

 **Create/edit polyline**

Create/edit plane



This is a toolbox for creating/editing a plane from points, axes, objects, UCS directions, etc.

Start by dragging and dropping in the editor window the plane you want to edit. If no plane is dropped, a new plane will be created when you activate any of the functions or when you click on .

Fitting

Several criteria are defined to fit the plane.

Creating a plane from two points

This option allows you to create a vertical plane from - or to make your plane pass by - two points. To define one of these points, just click on and double click on any point in the 3D scene. Otherwise, select any point from the point list window by clicking on... . It is also possible manually insert the coordinates of the points in the dedicated boxes. When you have defined the points, click on *Fit plane.* The plane will be adjusted to pass by the specified points, to be vertical (coplanar to the Z axis of the current UCS) leaving as much as possible unchanged the other plane's properties.

Creating a plane from video selection

This option allows you to create a plane from a video selection of point clouds.
Click on *Quick selection* to use the Lasso tool to make a free hand selection in the point cloud view. Automatically the plane will be created, fitting the selected points.
Click the *Fit plane to all visible points* button to create a plane fitting all the points visible on the 3D view (in the foreground).

Creating a plane from point list

This option allows you to create (or adapt) a plane from the point listed in the project.

Creating a plane from segment and point

This option allows you to create a plane from a segment and a point included in the project.
Select the segment and the point in the Project Window then drag&drop them in the provided boxes.
Two modalities are available to fit the plane:

- *The chosen segment is the normal plane*: the plane will be perpendicular to the segment
- *The chosen segment must lie on the plane*: the plane will have the segment lying on it and will pass by the point

After choosing the modality and the segment/point entities, click the *Make plane from segment and point* button to create/modify the plane.

Creating a plane from origin and axes

This option allows you to create a plane from - or to make your plane pass by - three points. These points are the center of the plane (UCS origin), the X (or Y) axis endpoint, and a point on XY plane. To define one of these points, just click on and double click on any point in the 3D scene. Otherwise, select any point from the point list window by clicking on... . It is also possible manually insert the coordinates of the points in the dedicated boxes. When you have defined the point(s), click on *Fit plane.* The plane will be adjusted to pass by the specified points, leaving as much as possible unchanged the other plane's properties.

Creating a plane from bounding box

This option allows you to create a plane from the faces of the cumulative bounding box of the currently selected project items. This is useful for example to immediately create the base plane of a certain church or cave, to make plants or sections later.
When you create/edit a plane from the bounding box of project item(s) (bottom group box), the plane's direction is determined by the current UCS: *frontal*, *lateral* and *base* plane mean that the plane's normal is parallel to the UCS' X, Y and Z axis respectively.

For some modalities, a default set of fitting parameters can be adopted or modified to interpolate the points.
Among the selected points, the RANSAC algorithm randomly selects X points and tries to fit the

geometric shape on them.
The algorithm then iterates this procedure N times keeping the best result that it found among all attempts. The quality of the fitting is given by how many points (out of the selected ones) are sufficiently close - and therefore belonging - to the geometric shape (*inliers*).
The maximum value of X is given by the *Maximum points used for fitting [#]* parameter.
The *RANSAC minimum iterations [#]* parameter defines the minimum number of attempts N that the algorithm must make before considering the geometric shape correctly fitted.
Inliers maximum distance [m] defines the maximum distance within which a point is considered to belong to the geometric shape.

Editing

Once the plane is defined, it can be edited by setting the dimensions along X and Y axes.
It can be also oriented according to the vertical horizontal directions (respectively parallel to Z axis and XY plane), as well as its X axis can be forced to be horizontal.

Duplicate plane as

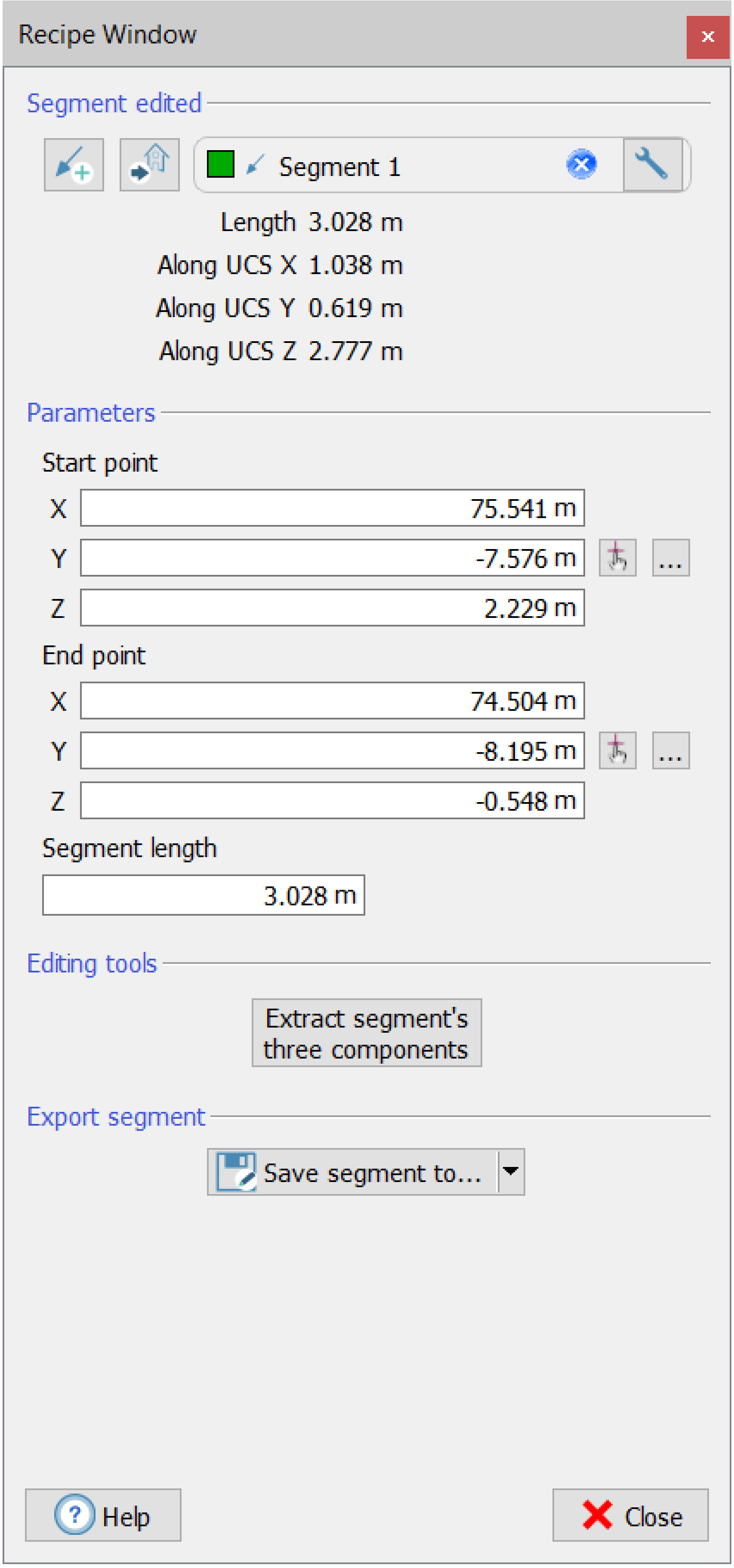
The selected plane can be duplicated as a

- *New plane*: a new and identical plane will be added to the project
- *New Orthocamera*: a new orthocamera in which the projecting plane will be the same plane will be added to the project.

  This option is useful when you want to extract a cross section and an orthophoto (or X-ray orthophoto) from the same position: create the cutting plane to generate the cross section, duplicate it as an orthocamera and use it to generate an orthophoto from the same point of view.

The functions explained before are more powerful if used in combination.
For example, first you create a plane from three points of a façade, then you make the plane vertical, or its X axis horizontal, to create later an orthocamera from it. Note also that, while you edit your plane, you have the undo/redo buttons on the bottom of the dialog, to make the editing process easier.
When you are done, just close the window. If you want to start a new plane, click on *I'm done, create a new plane.*

Create/edit segment

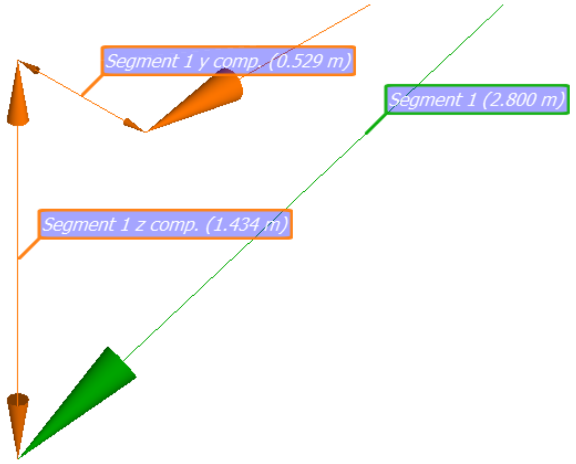


This is a toolbox for creating/editing a segment from a starting and an ending points.
Start by dragging and dropping in the editor window the segment you want to edit. If no segment is dropped, a new segment will be created when you click on .

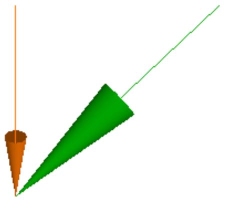
Note that a segment could be created also as intersection between other shapes (see Intersect geometric shapes).

To define one of these points, just click on and double click on any point in the 3D scene.

Otherwise, select any point from the point list window by clicking on... . It is also possible manually insert the coordinates of the points in the dedicated boxes. When you have defined the points, the segment (with arrows) will appear in the 3d view, as well as its length's value in the dialog.



By clicking on *Extract segment's three components* the components along X, Y and Z directions (of the current UCS) will displayed in the 3D view.

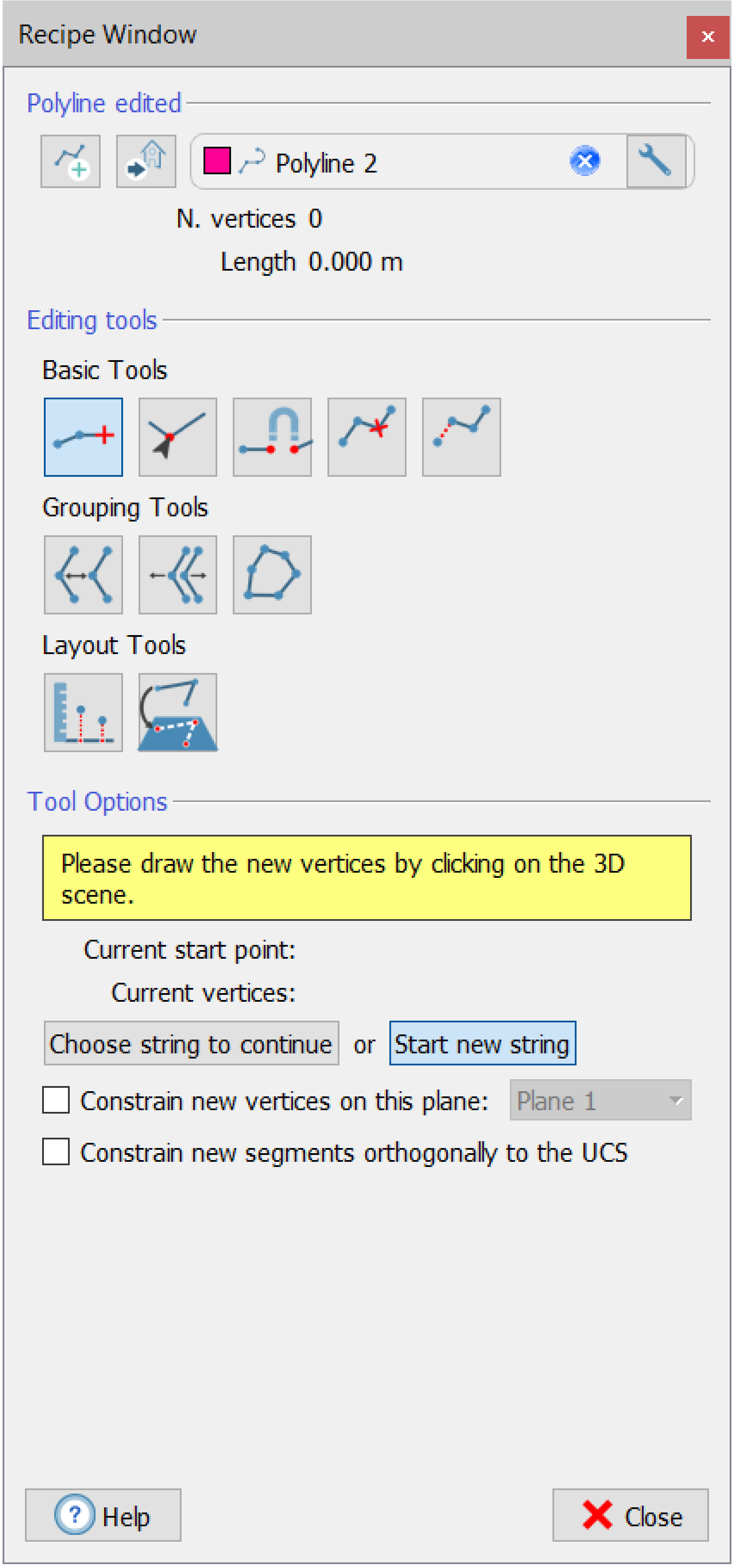


The *Save segment to...* command allows you to:

- copy to a new segment
- save the segment's information (below illustrated with an example), in .txt, .csv formats and as a clipboard.

```
Segment name: Segment 1
Length: 2.6511 m
Along UCS X: 0.3782 m
Along UCS Y: 2.5893 m
Along UCS Z: 0.4256 m
Start point coordinates: 12.0990, 3.1179, 2.0540 m
End point coordinates: 11.7208, 5.7072, 1.6284 m.
```

Create/edit polyline



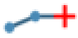
This is a toolbox for creating/editing a polyline by managing points in the 3D view.

Start by dragging and dropping in the editor window the polyline you want to edit. If no polyline is dropped, a new polyline will be created when you click on .

In the upper part of the window the number or vertices added to the polyline and its complete length are listed.

Editing Tools and Tools Options

Basic Tools



Add vertices

By clicking on the 3D scene, new vertices are added to the polyline. Doing it, the number of vertices and the length are automatically updated.

During the picking point process it is possible to include different strings and modify them in the polyline, by starting from one endpoint.
By selecting *Choose string to continue* and picking an endpoint in the 3D scene you can restart to add vertices from this position, whereas by selecting Start new string and picking points in the 3D scene, these points will be included in a new string.

Check *Constrain new vertices on this plane* and select a plane in the project to lie the picked points on it. The vertices added from this point will lie on the plane.
Check *Constrain new segments orthogonally to the UCS* to orient the new added segments as parallel to X, Y or Z axis of the current UCS. The segments added from this point will be only vertical (parallel to Z) or horizontal (parallel to X or parallel to Y).

Edit vertex

To select a vertex of the polyline in the 3D space and edit it by changing its position.

Connect vertices

To connect two vertices with a segment.
The vertices must be endpoints of strings, and can be selected by double clicking on them in sequence: first the First Vertex and then the Second Vertex.

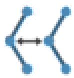
Remove vertex

To remove a vertex from the polyline, by double clicking on a point close to the vertex.

Remove segment

To remove segments by double clicking on them.

Grouping Tools



Merge polylines into one

To merge the current polyline with another one by double clicking on it in the 3D scene. At the end only one polyline will stay in the project, the current one.

Explode polyline strings

To explode a polyline string by double clicking on the string of the current polyline to explode. Any other strings will be moved to separate polylines.

Close polyline

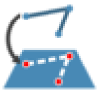
To close a polyline by double clicking on a string of a current polyline to close.

These tools can be used to redraw the point cloud in order to extract the desired main features as polylines.

Layout Tools

Set same altitude to all vertices

To set a constant altitude to the polyline's vertices. The altitude can be defined by picking a point in the 3D scene (Z value) or manually set in the specific box. Once the altitude is selected, click on *Set Altitude* to actualize the process.



Project polyline on plane
To project the polyline on a plane. The plane can be double clicked in the 3D space or selected from the list. Once the plane is selected, click on *Project* to actualize the projection.

Second order shapes

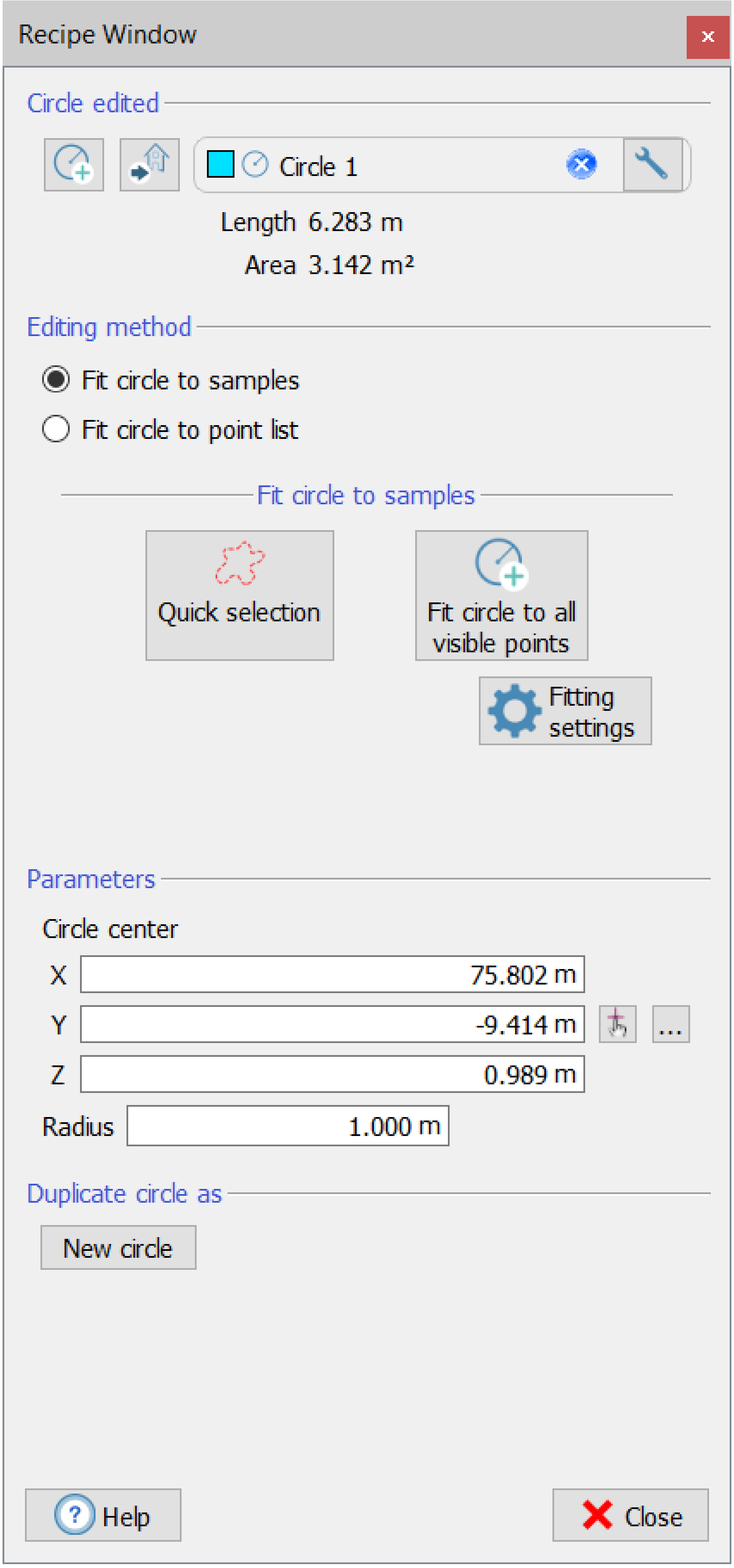
 **Create/edit circle**

 **Create/edit sphere**

 **Create/edit cylinder**

 **Create/edit cone**

Create/edit circle



This is a toolbox for creating/editing a circle from samples or point list.
Start by dragging and dropping in the editor window the circle you want to edit. If no circle is dropped, a new circle will be created when you activate any of the functions or when you click on .

Note that a circle could be created also as intersection between other shapes (see Intersect geometric shapes).

In the upper part of the window the length and the circumscribed area are listed.

Editing method

Several criteria are defined to fit the cylinder.

Fit cylinder to samples

This option allows you to create a cylinder from a video selection of point clouds.
Click on *Quick selection* to use the Lasso tool to make a free hand selection in the point cloud view.
Automatically the cylinder will be created/adapted, fitting the selected points.
Click the *Fit cylinder* button to create a circle fitting all the points in the 3D view.

Fit cylinder to point list

This option allows you to create (or adapt) a circle from the point listed in the project.

A default set of fitting parameters can be adopted or modified to interpolate the points.
Among the selected points, the RANSAC algorithm randomly selects X points and tries to fit the geometric shape on them.
The algorithm then iterates this procedure N times keeping the best result that it found among all attempts. The quality of the fitting is given by how many points (out of the selected ones) are sufficiently close - and therefore belonging - to the geometric shape (*inliers*).
The maximum value of X is given by the *Maximum points used for fitting [#]* parameter.
The *RANSAC minimum iterations [#]* parameter defines the minimum number of attempts N that the algorithm must make before considering the geometric shape correctly fitted.
Inliers maximum distance [m] defines the maximum distance within which a point is considered to belong to the geometric shape.

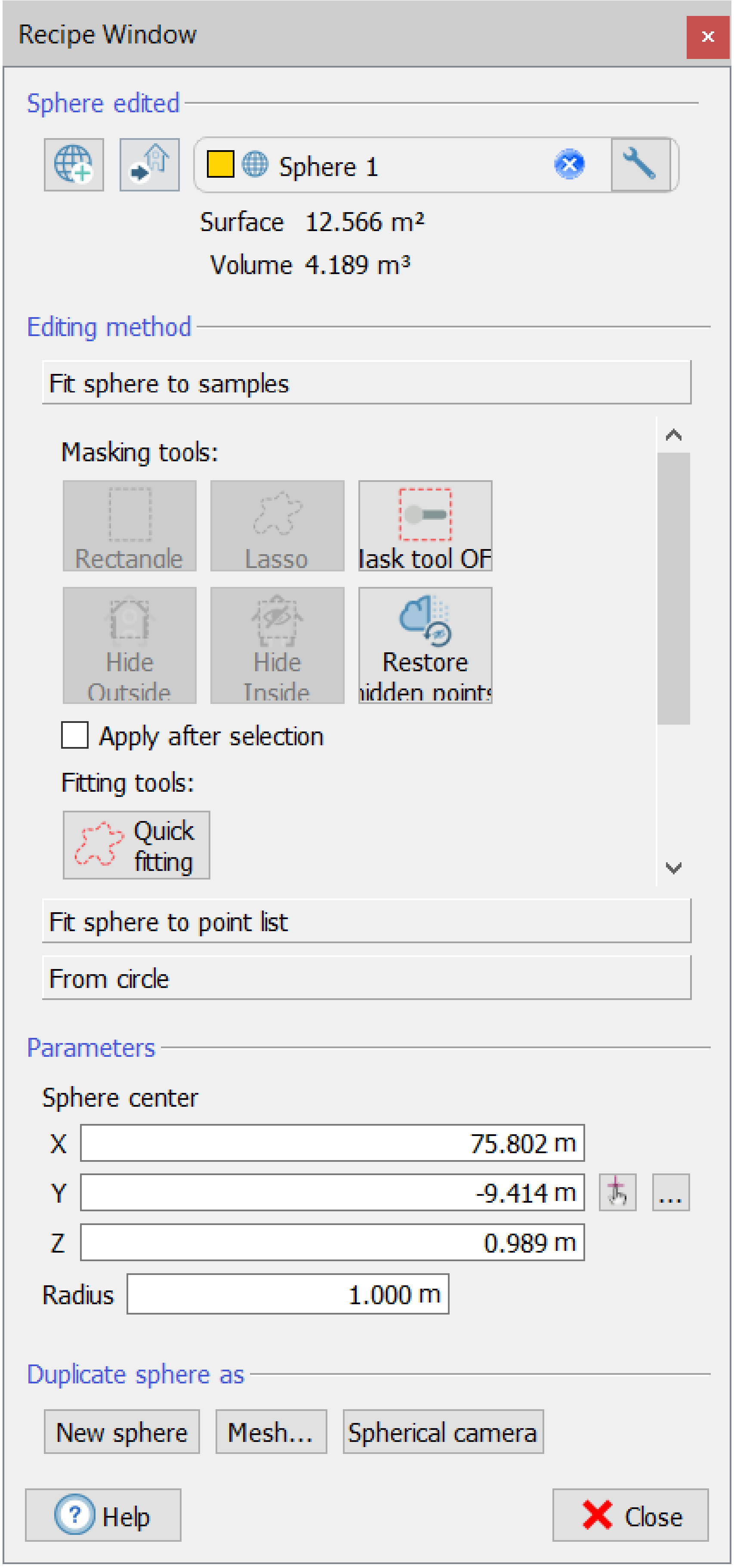
Parameters

Once the circle is defined, it can be edited by setting its center's coordinates and radius.

Duplicate circle as

The selected circle can be duplicated as a new circle: a new and identical circle will be added to the project.

Create/edit sphere



This is a toolbox for creating/editing a sphere from samples, points or circles.
Start by dragging and dropping in the editor window the sphere you want to edit. If no sphere is dropped, a new sphere will be created when when you activate any of the functions or when you click on .

In the upper part of the window the Surface and the circumscribed Volume are listed.

Editing method

Several criteria are defined to fit the sphere.

Fit sphere to samples

This option allows you to create a sphere from a video selection of point clouds.
Flag the *Fitted sphere must have radius* and insert a value for the radius of the fitting sphere if you know it .
Click on *Quick selection* to use the Lasso tool to make a free hand selection in the point cloud view. Automatically the sphere will be created/adapted, fitting the selected points.

The *Masking Tools* can be used to isolate small areas to look for spheres.

- Turn ON/OFF the Mask tool to select the points with Rectangle or Lasso tools
- Hide or Unhide points in the 3D view to better identify the correct selection
- Check "Apply after selection to update the visualization in the 3D view

Fit sphere to point list

This option allows you to create (or adapt) a sphere from the point listed in the project.

A default set of fitting parameters can be adopted or modified to interpolate the points.
Among the selected points, the RANSAC algorithm randomly selects X points and tries to fit the geometric shape on them.
The algorithm then iterates this procedure N times keeping the best result that it found among all attempts. The quality of the fitting is given by how many points (out of the selected ones) are sufficiently close - and therefore belonging - to the geometric shape (*inliers*).
The maximum value of X is given by the *Maximum points used for fitting [#]* parameter.
The *RANSAC minimum iterations [#]* parameter defines the minimum number of attempts N that the algorithm must make before considering the geometric shape correctly fitted.
Inliers maximum distance [m] defines the maximum distance within which a point is considered to belong to the geometric shape.

From circle

This option allows you to create (or adapt) a sphere from a circle listed in the project, used as Equator circle.

Parameters

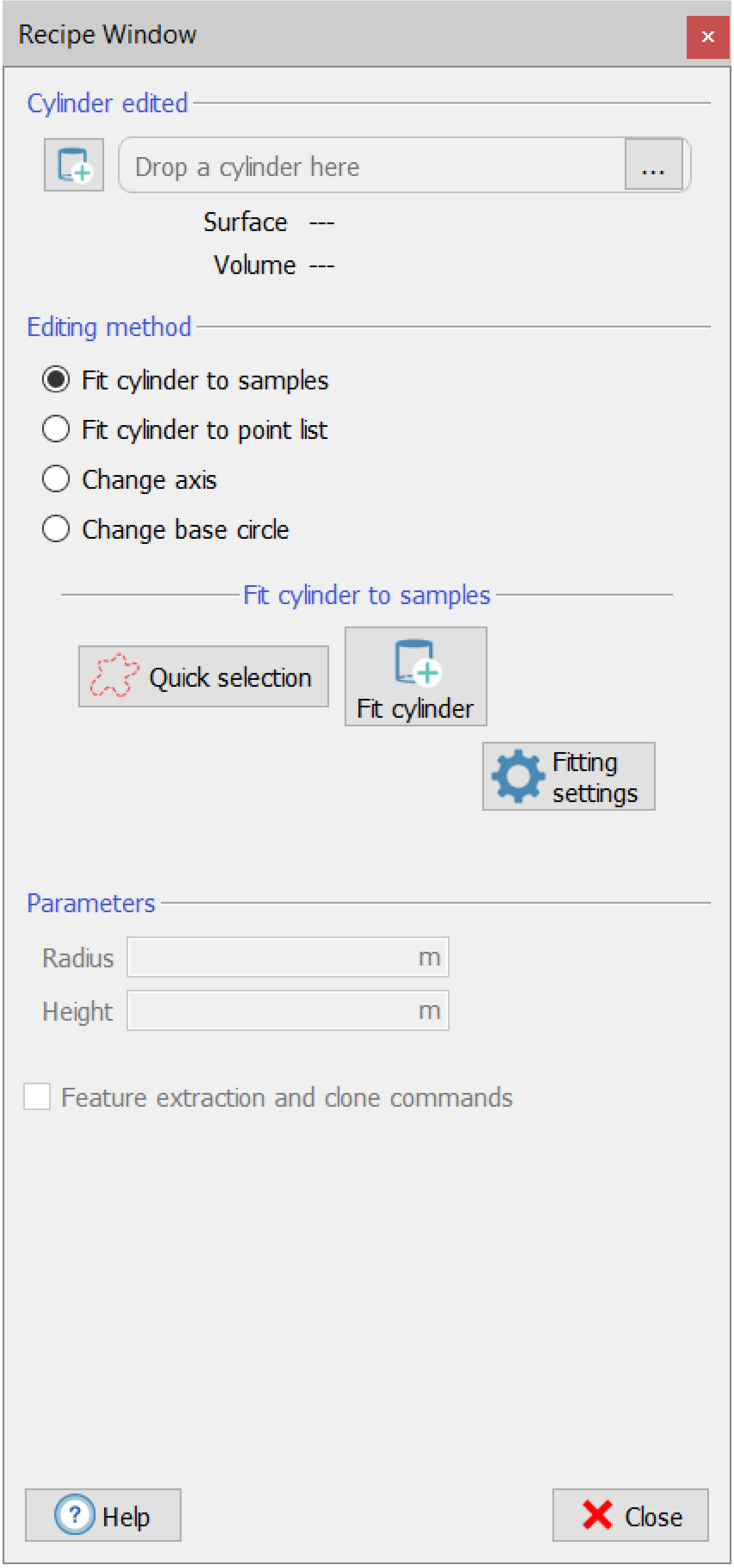
Once the sphere is defined, it can be edited by setting its center's coordinates and radius.

Duplicate circle as

The selected circle can be duplicated as

- a new sphere
- a Mesh
- a spherical camera.

Create/edit cylinder



This is a toolbox for creating/editing a cylinder from samples, points, axis or circles.
Start by dragging and dropping in the editor window the cylinder you want to edit. If no cylinder is dropped, a new cylinder will be created when when you activate any of the functions or when you click on  .

In the upper part of the window the Surface and the circumscribed Volume are listed.

Editing method

Several criteria are defined to fit the cylinder.

Fit cylinder to samples

This option allows you to create a cylinder from a video selection of point clouds.
Click on *Quick selection* to use the Lasso tool to make a free hand selection in the point cloud view.
Automatically the cylinder will be created/adapted, fitting the selected points.
Flag *Fit cylinder* to fit all the points in the 3D view.

Fit cylinder to point list

This option allows you to create (or adapt) a cylinder from the point listed in the project.

A default set of fitting parameters can be adopted or modified to interpolate the points.
Among the selected points, the RANSAC algorithm randomly selects X points and tries to fit the geometric shape on them.
The algorithm then iterates this procedure N times keeping the best result that it found among all attempts. The quality of the fitting is given by how many points (out of the selected ones) are sufficiently close - and therefore belonging - to the geometric shape (*inliers*).
The maximum value of X is given by the *Maximum points used for fitting [#]* parameter.
The *RANSAC minimum iterations [#]* parameter defines the minimum number of attempts N that the algorithm must make before considering the geometric shape correctly fitted.
Inliers maximum distance [m] defines the maximum distance within which a point is considered to belong to the geometric shape.

Change axis

This option allows you to create a cylinder by using as axis a segment (also distance) listed in the project.

Change a base circle

This option allows you to create a cylinder by using as base a circle listed in the project.

Parameters

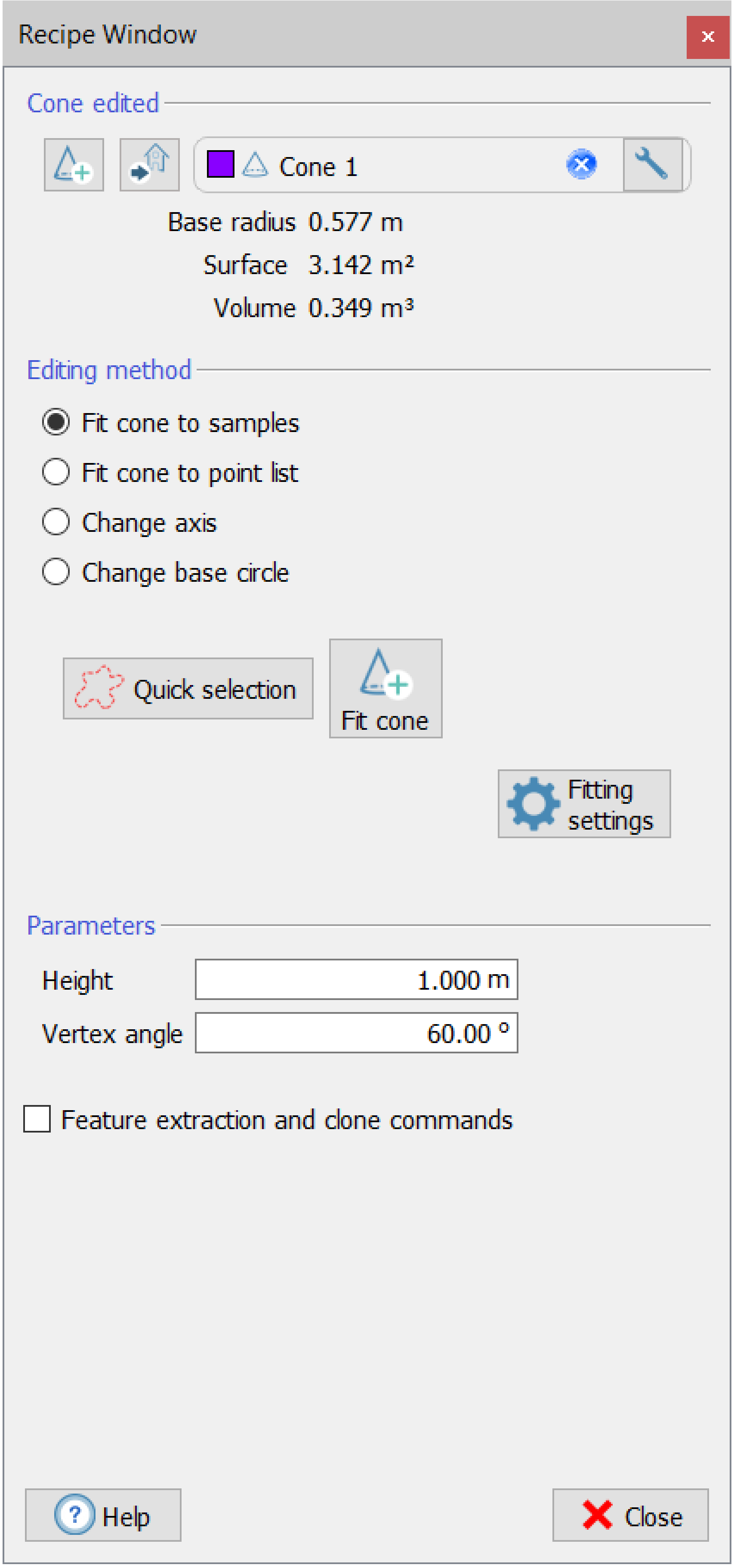
Once the cylinder is defined, it can be edited by setting its radius and height values.

Duplicate and extract

By flagging *Feature extraction and clone commands* some tools are available:

- Extract cylinder's axis
- Extract cylinder's top circle
- Extract cylinder's bottom circle
- Duplicate cone as a new cylinder
- Duplicate cone as a mesh
- Duplicate cone as a cylindrical camera

Create/edit cone



This is a toolbox for creating/editing a cone from samples, points, axis or circles. Start by dragging and dropping in the editor window the cone you want to edit. If no cone is dropped, a new cone will be created when when you activate any of the functions or when you click on .

In the upper part of the window the Base radius, the Surface and the circumscribed Volume are listed.

Editing method

Several criteria are defined to fit the cone.

Fit cone to samples

This option allows you to create a cone from a video selection of point clouds.
Click on *Quick selection* to use the Lasso tool to make a free hand selection in the point cloud view.
Automatically the cone will be created/adapted, fitting the selected points.
Flag *Fit cone* to fit all the points in the 3D view.

Fit cone to point list

This option allows you to create (or adapt) a cone from the point listed in the project.

A default set of fitting parameters can be adopted or modified to interpolate the points.
Among the selected points, the RANSAC algorithm randomly selects X points and tries to fit the geometric shape on them.
The algorithm then iterates this procedure N times keeping the best result that it found among all attempts. The quality of the fitting is given by how many points (out of the selected ones) are sufficiently close - and therefore belonging - to the geometric shape (*inliers*).
The maximum value of X is given by the *Maximum points used for fitting [#]* parameter.
The *RANSAC minimum iterations [#]* parameter defines the minimum number of attempts N that the algorithm must make before considering the geometric shape correctly fitted.
Inliers maximum distance [m] defines the maximum distance within which a point is considered to belong to the geometric shape.

Change axis

This option allows you to create a cone by using as axis a segment (also distance) listed in the project.

Change a base circle

This option allows you to create a cone by using as base a circle listed in the project.

Parameters

Once the cone is defined, it can be edited by setting its height and vertex angle values.

Duplicate and extract

By flagging *Feature extraction and clone commands* some tools are available:

- Extract cone's axis
- Extract cone's base circle
- Duplicate cone as a new cone
- Duplicate cone as a mesh
- Duplicate cone as a perspective camera

Shapes interactions

Distance between shapes

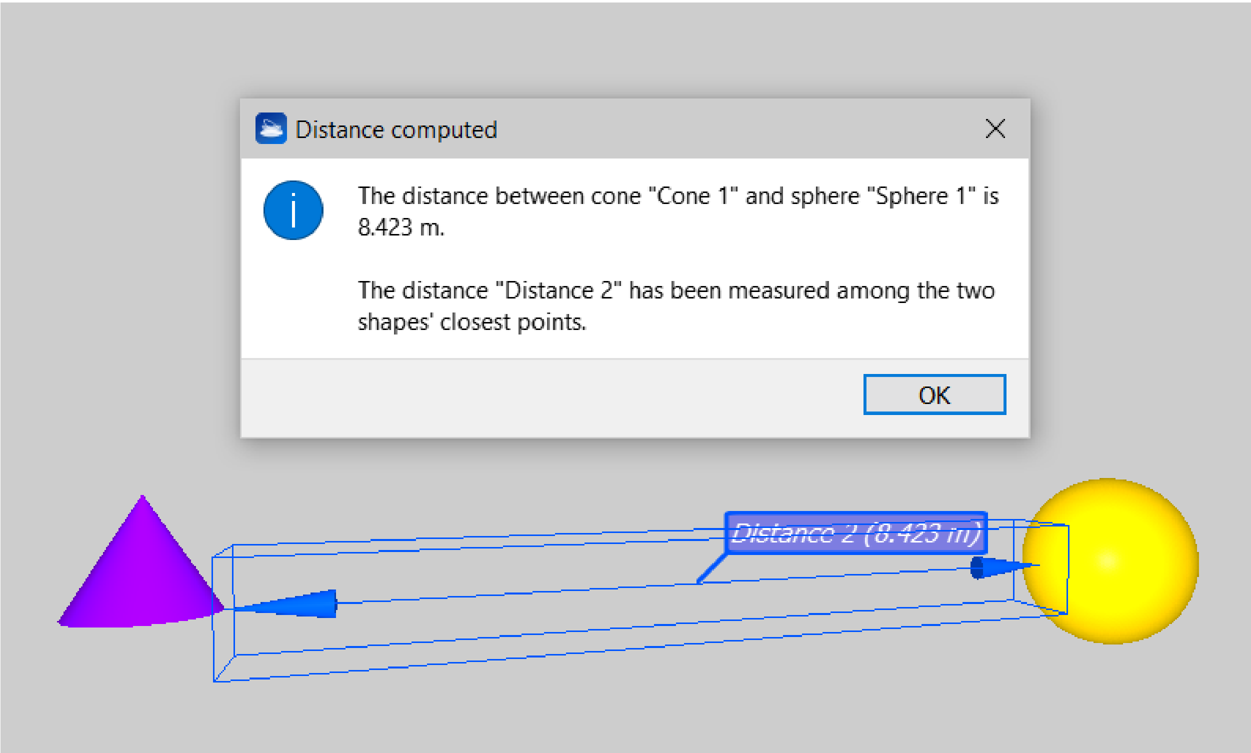
Intersect geometric shapes

These commands are available in the *Mesh&Shapes* Top Toolbar and in the *Data Management* > *Geometric shapes* Top menu.

By directly selecting two shapes in the Project Window and right-clicking with the mouse, a context menu allows you to select one of these two tools and obtain the distance or the intersection between the selected shapes in a fast way.

Distance between shapes

Selected two shapes, the distance among the two shapes' closest point is computed and inserted in the *Distances* items in the project.



If shapes intersect, no distance is computed.

See also Distance tool.

Intersect geometric shapes

Selected two shapes, the intersection among the two shapes is computed and inserted in the *Geometric shapes* items in the project.
The extracted geometric shape depends on the starting shapes and on their relative position.
It could be a segment, a circle or a point.

For example, the intersection between a plane and a sphere is a circle (or a point if they are tangent):

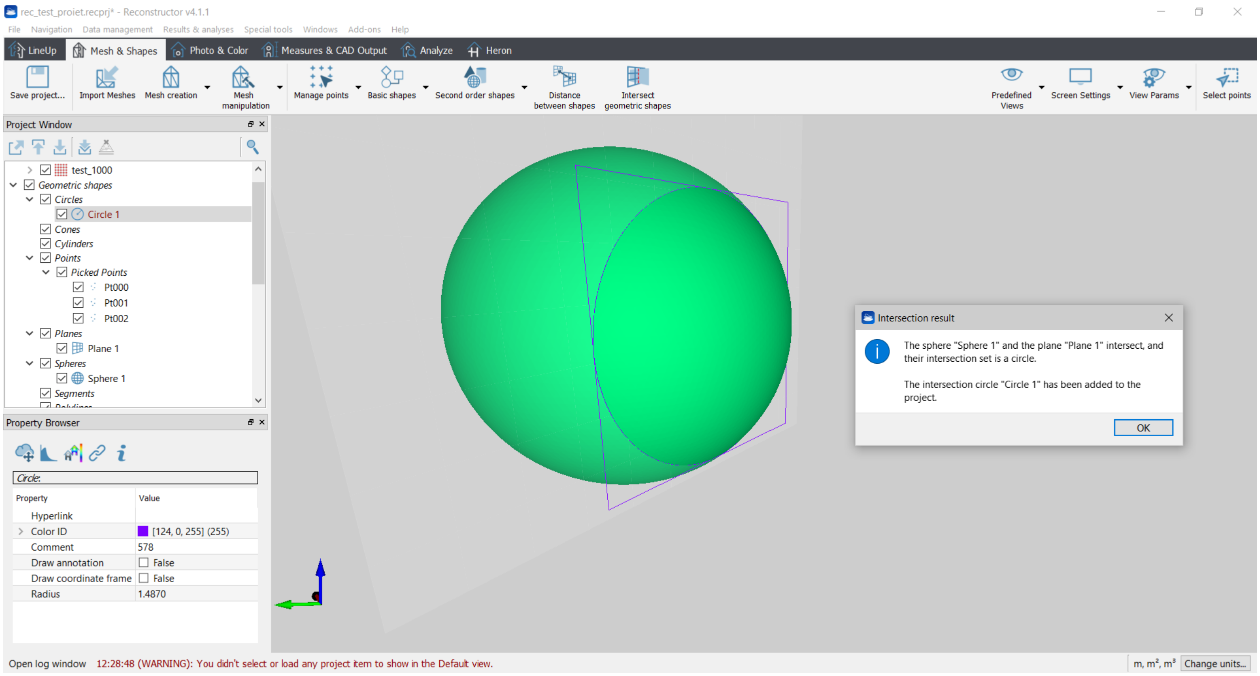


Photo & Color

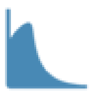
Reconstructor® provides several coloring tools to effectively manage the colorization of point clouds and meshes.

These tools can be classified according to the objects you want to color and the procedures you need to apply.

Point clouds' color layers managing

Some color information is item intrinsic of point clouds and saved as a color layer, in addition of the ones already included in the raw data (i.e. Reflectance, Color, ...) or after pre-processing step (i.e. Confidence, Inclination, ...).

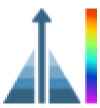
By using the tools below listed it's possible to create and add new layers of color.



Histogram

To optimize the point clouds color's contrast by histogram stretching. Only single channel layer can be used (reflectance, gray scale layers...).

This command is available in the *Property browser* dialog.



Color with altitudes

To add to the selected point cloud(s) an extra color layer, representing the altitude of the points with respect to one of the tree axes of the current UCS.

Inclination from plane
To add to the selected point cloud(s) an extra color layer, representing the inclination of the points' normals with respect to a given plane that exists in the project.

Use it to classify points for a further vegetation removal.

Color with range
To add a layer coloring the distance from the relative observation point.

Color with ambient light
To add to the selected point cloud(s) an extra color layer,

Visualization of outdoor large mapped areas can be enhanced with this layer.

Compute 3D reflectance
To add to the selected point cloud(s) a special color layer, named *3D Reflectance*, that mixes LiDAR intensity and point normals (*inclination*) to enhance both geometrical and surface characteristics.

Add color layer from image
To apply an image on the point cloud (e.g. a 2Dview after a color editing).

Pay attention to the dimensions of the images: they must be the same of the point cloud ones.

Merge or split layers
To combine RGB and scalar layers to obtain a customized new layer of color for point clouds.

A particular and useful tool is dedicated to manipulation of layers of colors:

Colors mapping
To manage an artificial colorization for a given *color layer* of a point cloud. Color information can be scalar (e.g. reflectance, range, confidence) or vectorial (e.g. inclination). It's also possible to add layers from external images. The purpose of this command is to manage and create an artificial colorization for a given *color layer* of a point cloud.

Camera calibration

Images from external cameras can be calibrated (oriented) on point cloud and meshes to obtain colored 3d models (see at the Colorization paragraph).

Images & calibration
This tools are used to calibrate images on point clouds and meshes and to determine the external and intrinsic parameters of a camera:

Camera calibration on a model

Camera calibration using 3D points

Calibrate camera intrinsic

Manage projectors

Several kind of 3D projectors can be created in Reconstructor® and used to project images on the 3D scene.

Create projector

This functions creates a projector in four different frustum types: Perspective, Orthographic, Spherical and Cylindrical projectors. An additional way to create perspective or spherical projectors is done by a calibration (by importing a Reconstructor® camera calibration file) using a valid image.

Load projector

To load (light on) the image on the 3D model.

Unload projector

To unload the image from the 3D model.

Colorization

A mapping (on point clouds and meshes) of calibrated images from external cameras is possible, in order to obtain colored point cloud and meshes.

Texture mapping

To color point clouds and meshes by using external images (and also extracted panoramic cameras from perspective picture in Reconstructor®)

Color clouds with photos

Create panorama

To create a create panorama (spherical 360° equi-rectangular) images starting from perspective images acquired with a camera centered on a 3D scanner head and rotating with them.

When you import .zfs or .fls point clouds a spherical camera (projector) can be automatically created (see LineUp import Wizard settings).

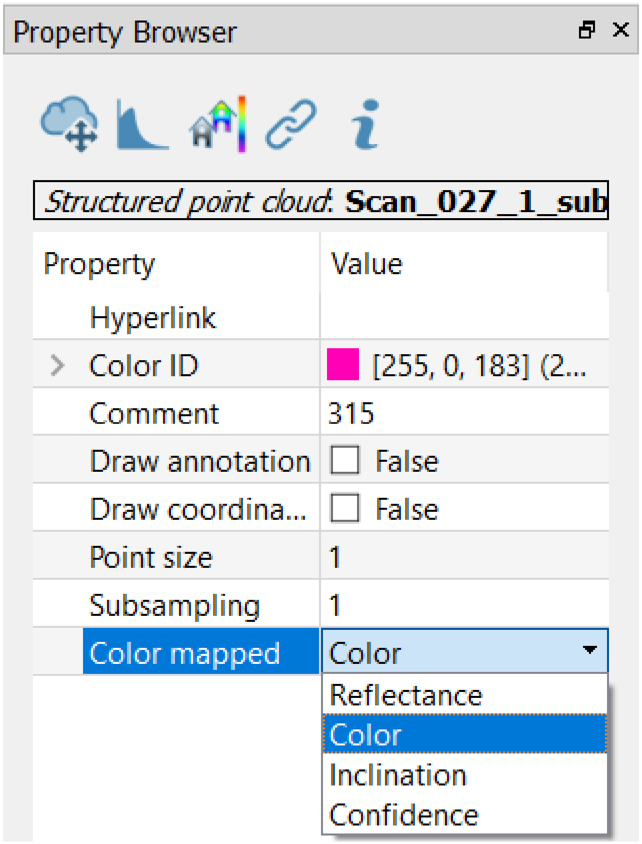
Create texture map

Opens a tool to texture map a selected mesh, given a texture view (a camera or a projector) and one or more projectors (calibrated images).

UAV and imaging

This tool is used to Import GeoTIFF as an orthographic camera.

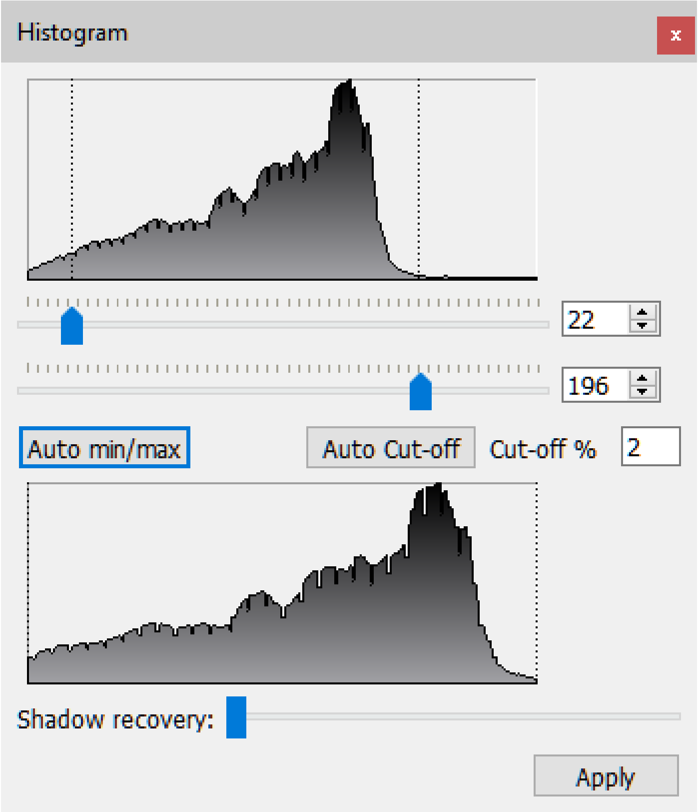
Import georeferenced tif...
Imports a georeferenced tiff (GeoTIFF) file as an orthographic camera.



To change the point clouds' color layers, select the point cloud from the project window and set the *Color Mapped* option in the Property Browser.

By opening the Colors mapping tool, a complete overview of the layers of color is available.

Histogram dialog



Given a point cloud and one of its scalar color layers (e.g. Reflectance), the histogram tool allows to optimize the color's contrast by histogram stretching. The histogram shown in the dialog represents how many points belong to a given color value.

This tool is available only if the cloud is loaded in memory and if the current color type is scalar, therefore it doesn't work on *Inclination* for example).
The two sliders allow you to decide the start and end of the histogram interval to be rendered.
The *Auto Cut-off* button allows you to automatically set the histogram minimum and maximum bin so that a certain percentage (2% in the figure above) of the histogram energy is discarded starting from zero to the minimum value and starting from the highest histogram bin back to the maximum value.

See also Point clouds and Colors mapping dialog.

Shadow recovery: lightens darker points

Colors mapping

Colors mapping
The purpose of this command is to manage and to create an artificial colorization for a given *color layer* of a point cloud.

The user can manage the rest of Reconstructor®'s GUI while keeping this dialog open.

When a point cloud is pre-processed, extra-information is added, and this is organized into *color layers*. Information can be scalar (e.g. reflectance, range, confidence, inspection) or vectorial (e.g. color, inclination).
When a color layer is rendered, a colorization is used to associate the points' color values to a drawable color. Only scalar information can be mapped with colors through this dialog. The color layer *Inclination* is an exception: it represents the normals of the points and it is colored automatically.

Only one color layer at a time is rendered in the 3D scene.

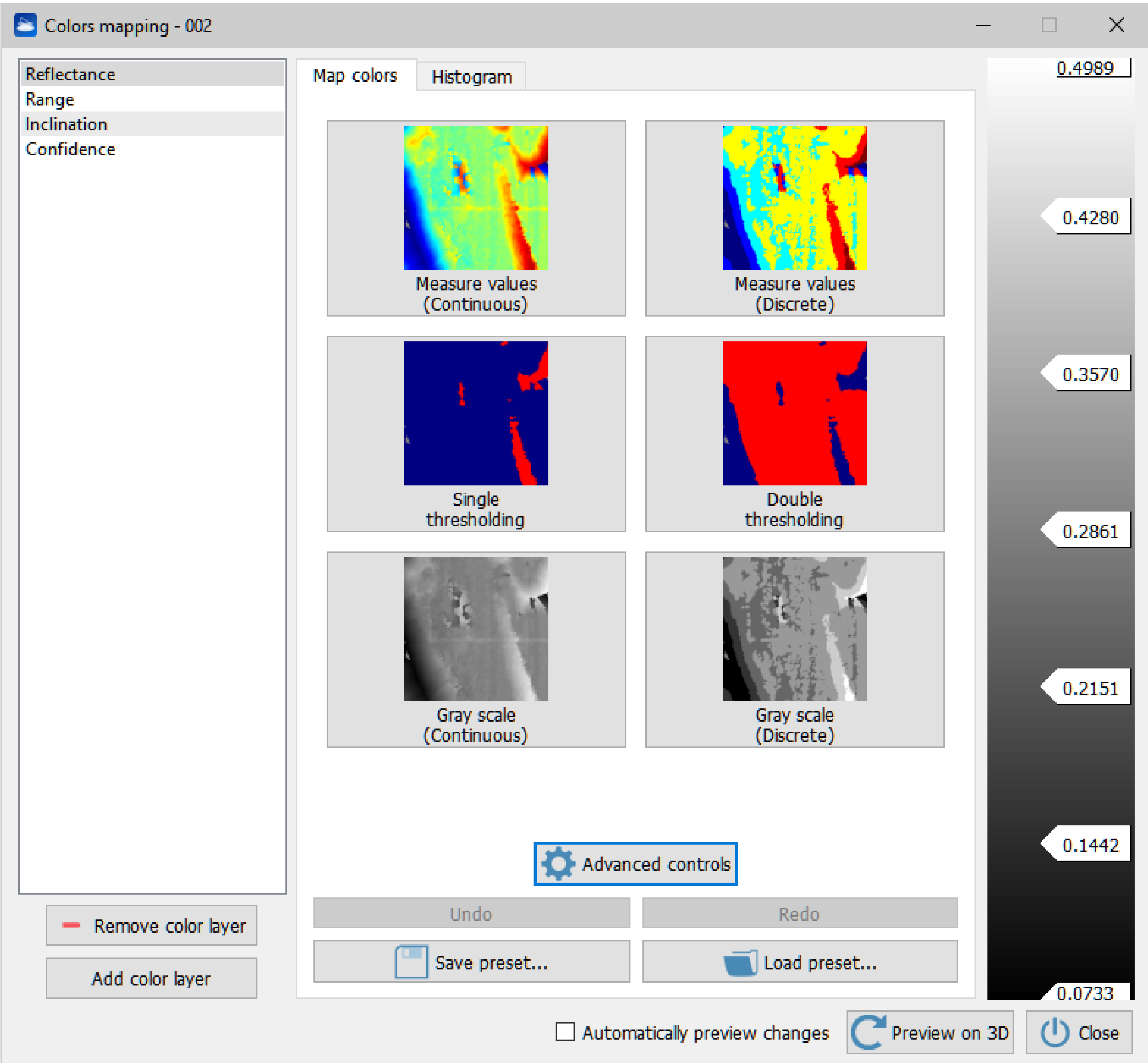
Layers list

On the top left of the Colors mapping dialog is a layers list showing which layers are associated to the current point cloud.
User can use it to switch from one color layer and another as well as can use the relative command in the property browser.

After selecting a cloud's color layer, the user can remove it by pressing the *Remove color layer* command.

Map colors tab

The Colors mapping window has the interface as below if a scalar layer is selected.



Presets

In the *Map Color* tab there is a set of buttons to apply a pre-defined colorization to the current selected layer.
Buttons used to map colors are:

- *Measure values (continuous)* : map the current layer to a continuous Rainbow Matlab style color.
- *Measure values* : map the current layer to a discrete Rainbow Matlab style color.

- *Single thresholding*: map the current layer to solid red and blue.
- User can set a threshold moving up and down the central value label.
- *Double thresholding*: map the current layer to solid red and blue.
  User can set two thresholds moving up and down the values labels.
- *Gray scale*: map the current layer to a continuous gray gradient.
- *Gray scale (discrete)*: map the current layer to a discrete gray gradient.

Color scale properties

On the right there is a color scale that shows the selected colors map in combination with labels. Labels show maximum and minimum (bounding) values and also values between colors. User can choose among several types of color scales via *Presets* and by managing the *Editing Options*. User can also change and disable colors by clicking on the color check box.

The following scale properties are displayed:

- *Max mapped* to set the max value and show it in the legend (click on *Reset* button to reset to the maximum absolute value)
- *Min mapped* to set the min value and show it in the legend (click on*Reset* button to reset to the maximum absolute value)
- *Colors scale interval with* that gives information about width of the color intervals
- *Unit type* that is depending on the type of colorization

Editing options

Colors intervals in the scale can be subdivided by:

- *Fixed intervals* according to the unit type (measure of unit)
- *Fixed bins* according to a defined number of equal spaced bins

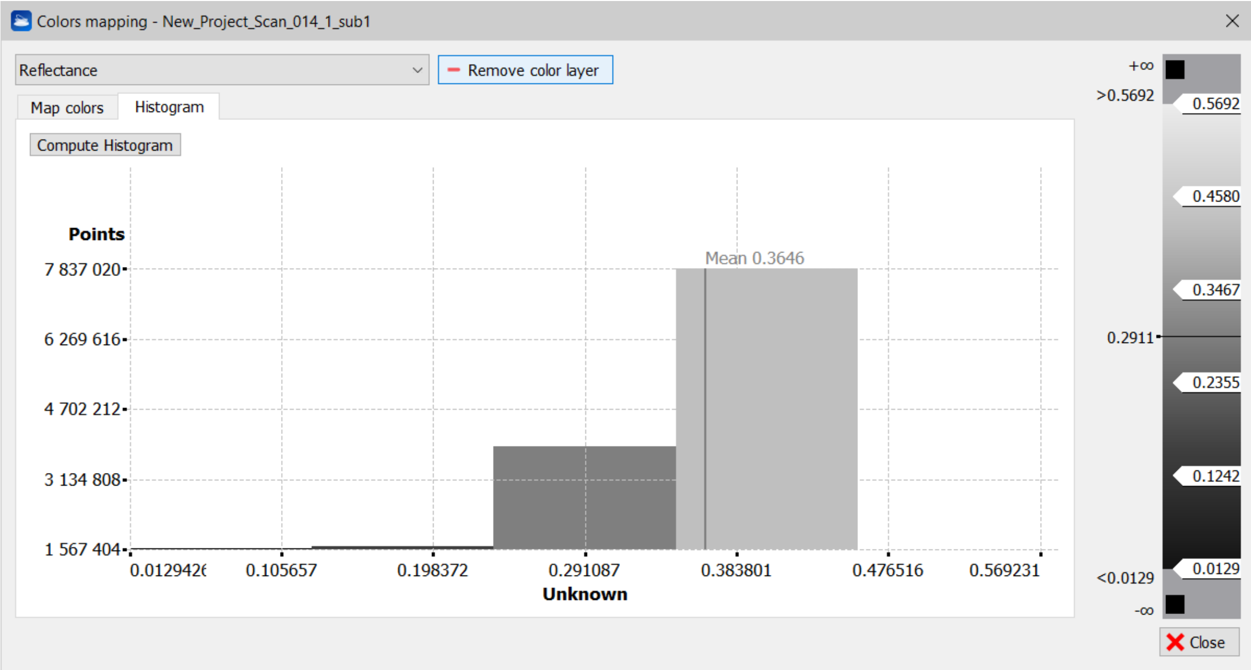
The scale type can be modified by selecting one of the following options:

- Extended
- Gray scale
- Rainbow
- Rainbow Matlab
- Ironbow

Setting for > Max, < Min and disabled intervals

When out of range, two modalities of points colorization can be chosen:
Color points: all the points out of the (Min,Max) interval are colored with an user defined color or with a chosen cloud layer of color (e.g. Reflectance, Inclination, ...)
Hide points: all the points out of the (Min,Max) interval are hidden.
By clicking on the *Unhide* button all the points previouvsly hidden will be made visible.

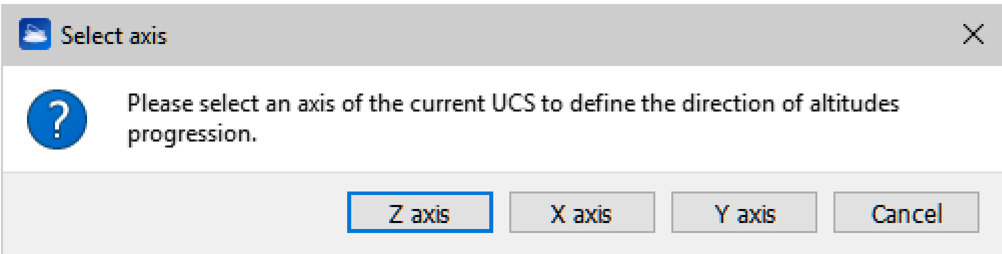
Histogram tab

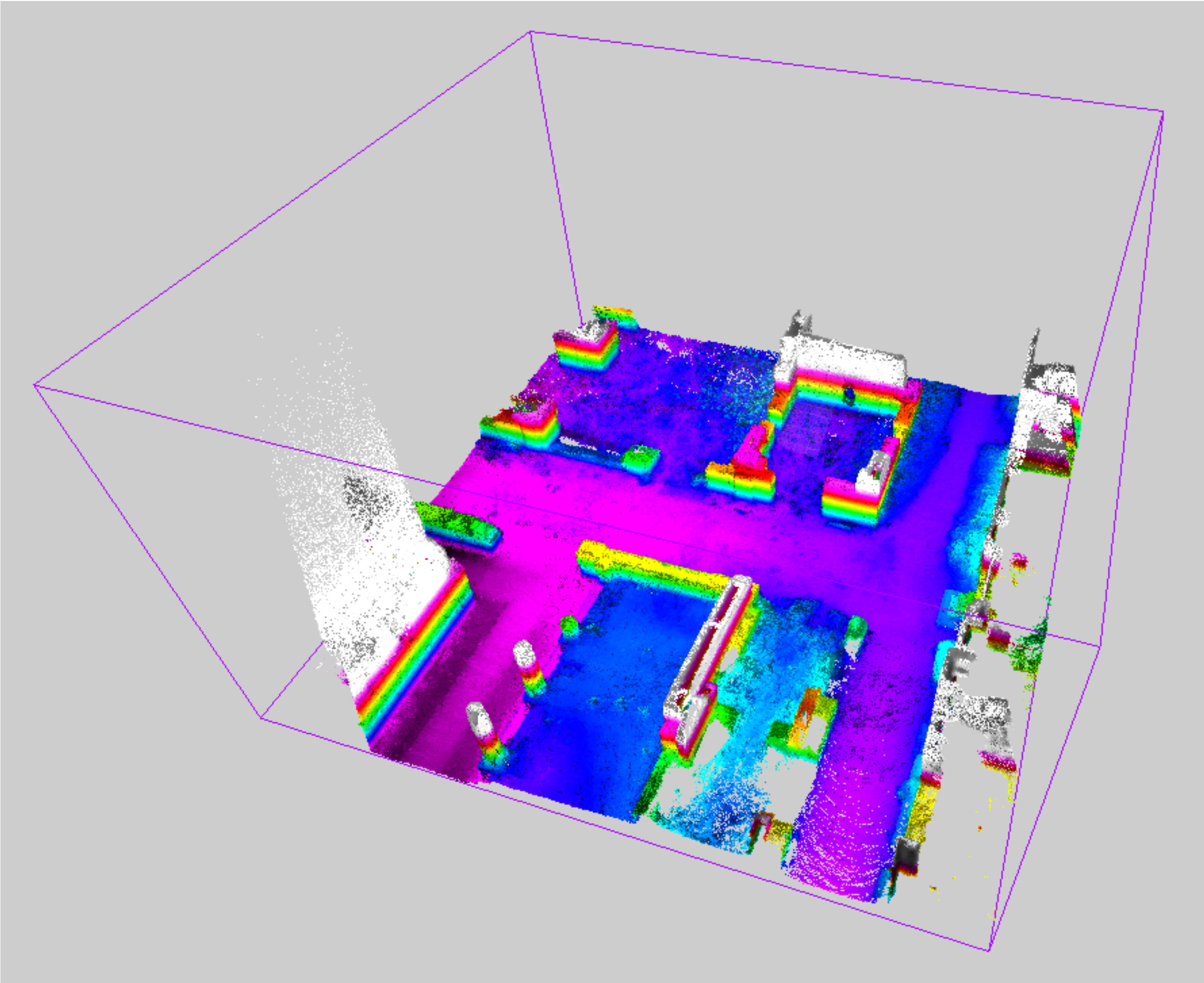


A colored histogram shows a statistical distribution with colors chosen by user. In abscissa, the histogram information about current measure (each color identifies a local radius of the measure) and in ordinate a measure of how many occurrences of points are mapped to a color. The mean value is also shown.
Click on *Compute Histogram* to extract it after creating or modifying it.

Color with altitude

This tool works on any set of point clouds. It allows to add to the selected cloud(s) an extra color layer, representing the altitude of the points with respect to one of the tree axes of the current UCS.





Change the color scale in the Color Mapping window.

Inclination from plane

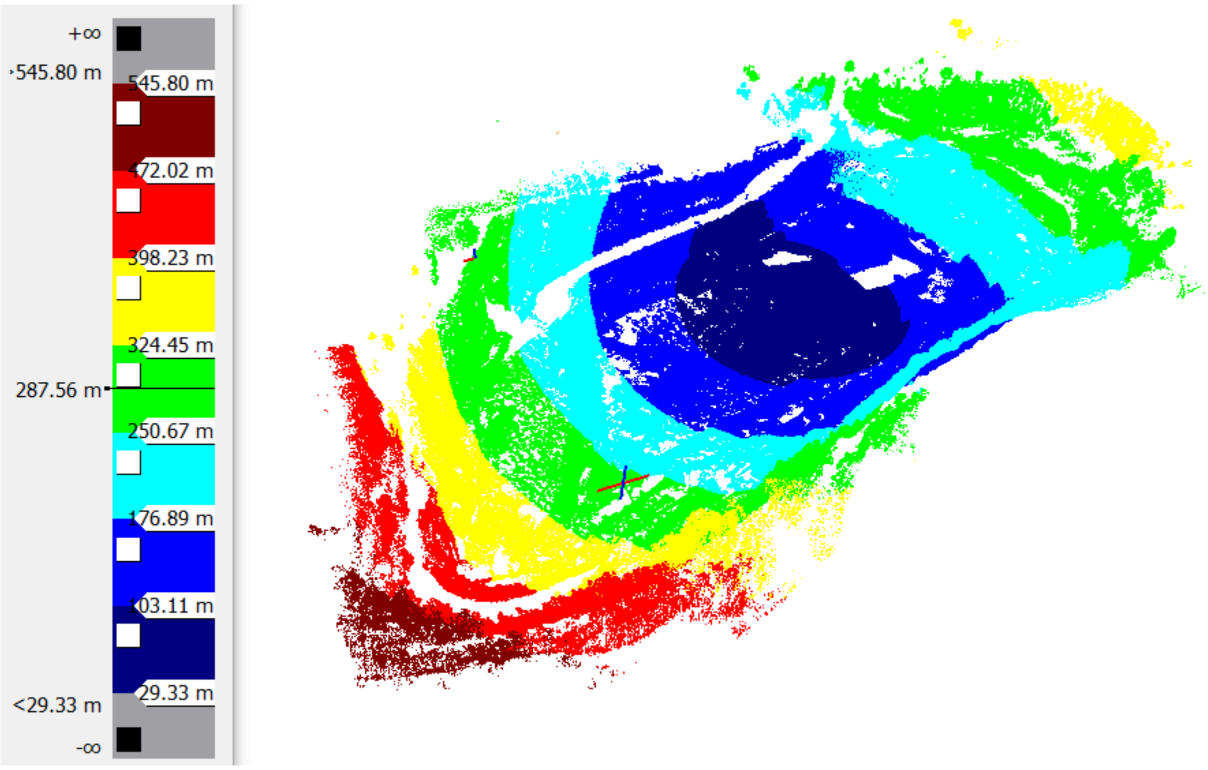
This tool works on any set of point clouds. It allows you to add to the selected cloud(s) an extra color layer, representing the inclination of the points' normals with respect to a given plane that exists in the project. The inclination from plane ranges from -1 to 1 and is calculated as the scalar product between the points' normals and the plane's normal.

It's possible to use this command, e.g., to distinguish points on vegetation to points on soil (with regular inclination) and remove them is an useful application of vegetation removing.

Color with range

This tool works on any set of point clouds. It allows to add to the selected cloud(s) an extra color layer, representing the distance of the points with respect to the origin of the point cloud.

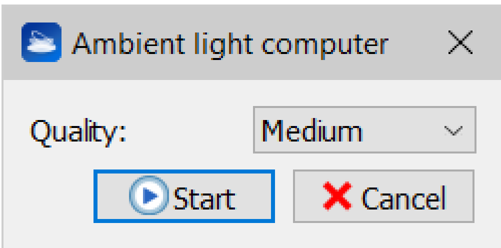
By default the layer's colorization is as in the image below, but can be modified with Colors Mapping tool.



Example of Color with Range colorization

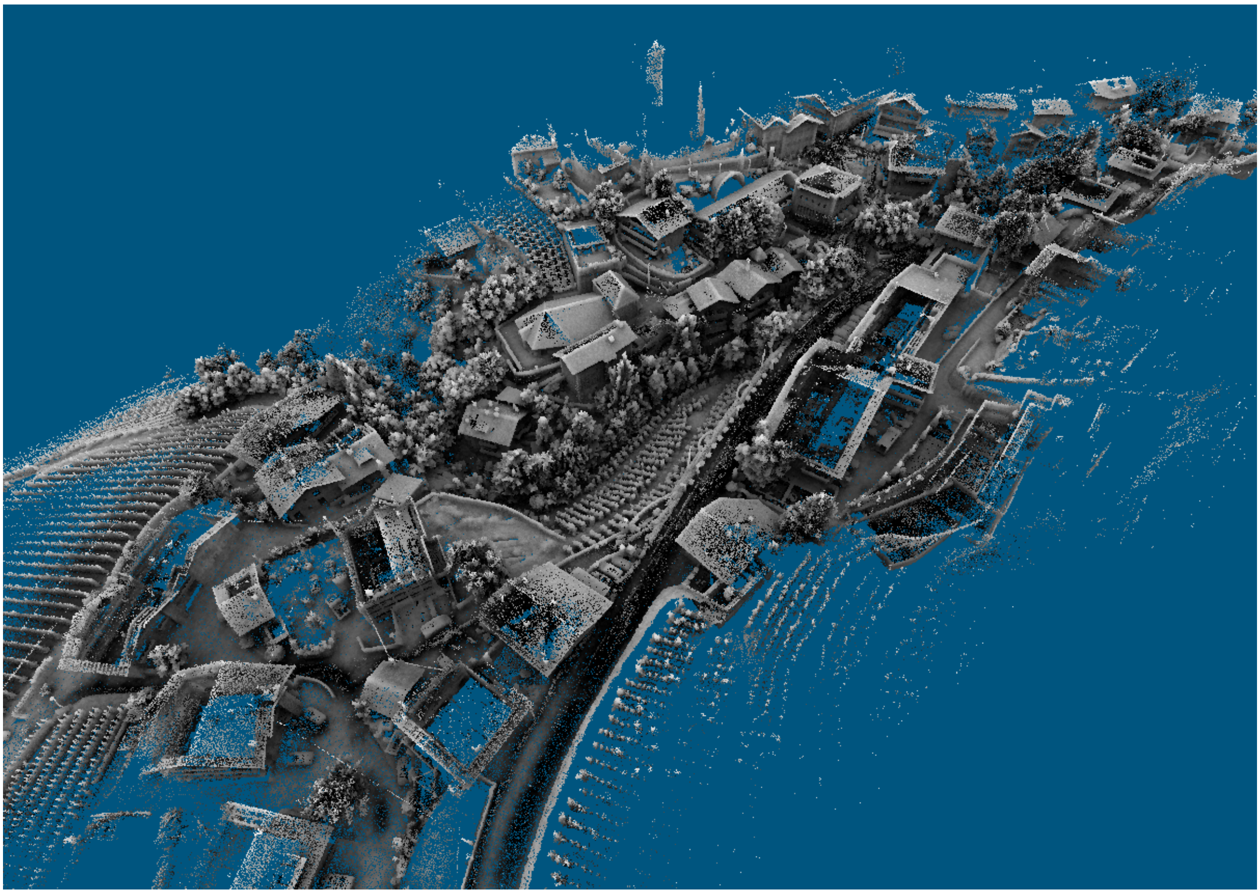
Color with ambient light

This tool works on any set of point clouds. It allows to add to the selected cloud(s) an extra color layer, named "AmbientLight", which simulates a light source that doesn't come from a particular direction. The light effect is so uniform throughout the scene.



Quality (Low, Medium or High) defines the deepness of the process: higher the quality, better the definition of the colorization and higher the computation time.

Visualization of outdoor large mapped areas can be enhanced with this layer.

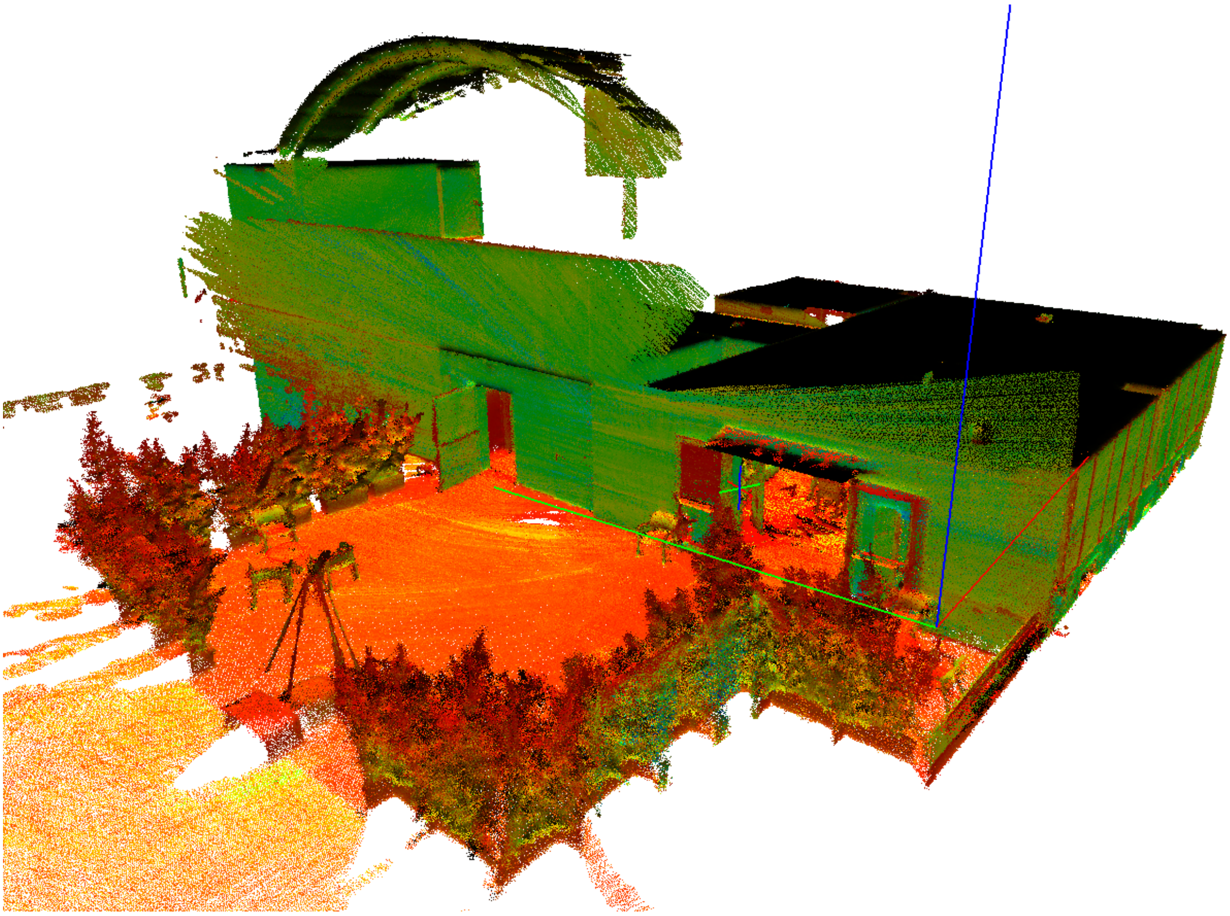


Example of Ambient Light colorization

Compute 3D reflectance

This tool works on any set of point clouds. It allows to add to the selected point cloud(s) a special color layer, named *3D Reflectance*, that mixes LiDAR intensity and point normals (*inclination*) to enhance both geometrical and surface characteristics.

Before computing the 3D Reflectance layer make sure that both *Reflectance* and *Inclination* layers are present in the point cloud(s).



Add color layer from image

This tool works only on structured point clouds (grids) and colors the point cloud by using any external image that can be projected on it.
Several formats of image are available (png, bmp, jpg, jpeg, tif, tiff) both for perspective and panoramic images.

This command is particularly used to modify the colorization of a scan by following this workflow:

- Open the contextual Edit 2D grid view
- In the *Point picking* section select the current color layer you want to manage and modify and click on *Save image* to save the current image to file.
- Open the image in a photo editor software and modify it
- Use the *Add color layer from image* tool to re-apply a new colorization to the scan loading the changed picture.

Pay attention to the dimensions of the images: they must be the same of the point cloud ones.

Merge or split layers

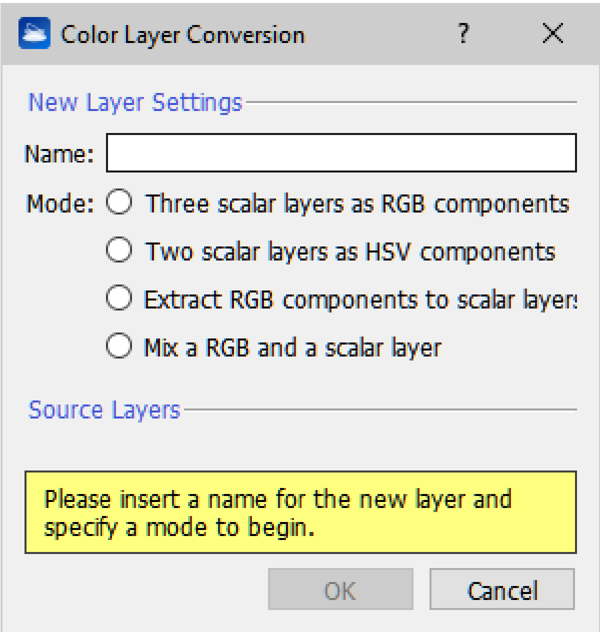
This tool works on any set of point clouds. It allows to combine RGB and scalar layers to obtain a customized new layer of color for point clouds.

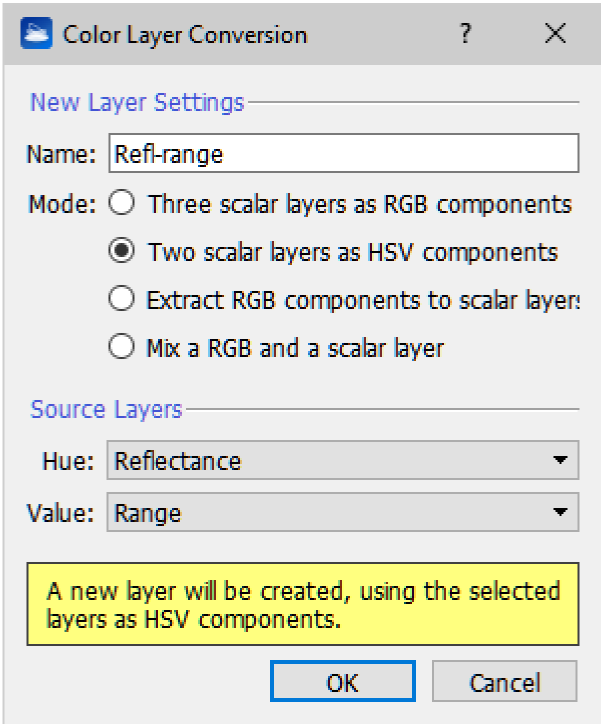
Color layers in Reconstructor can be distinguished in RGB layers and scalar layers.

RGB data holds the RGB (Red Green Blue) values that define the color of the points (as pixels in an image). Scalar data, on the other hand, defines properties of the colorization that can be used during computations. HSV (for hue, saturation, value; also known as HSB, for hue, saturation, brightness) are alternative representations of the RGB color model. In these models, colors of each hue are arranged in a radial slice, around a central axis of neutral colors which ranges from black at the bottom to white at the top.

Examples of RGB layers are *Inclination* and *Color*, while examples of scalar layers are *Reflectance*, *Confidence*, *Altitude*, *Inclination respect a plane*.

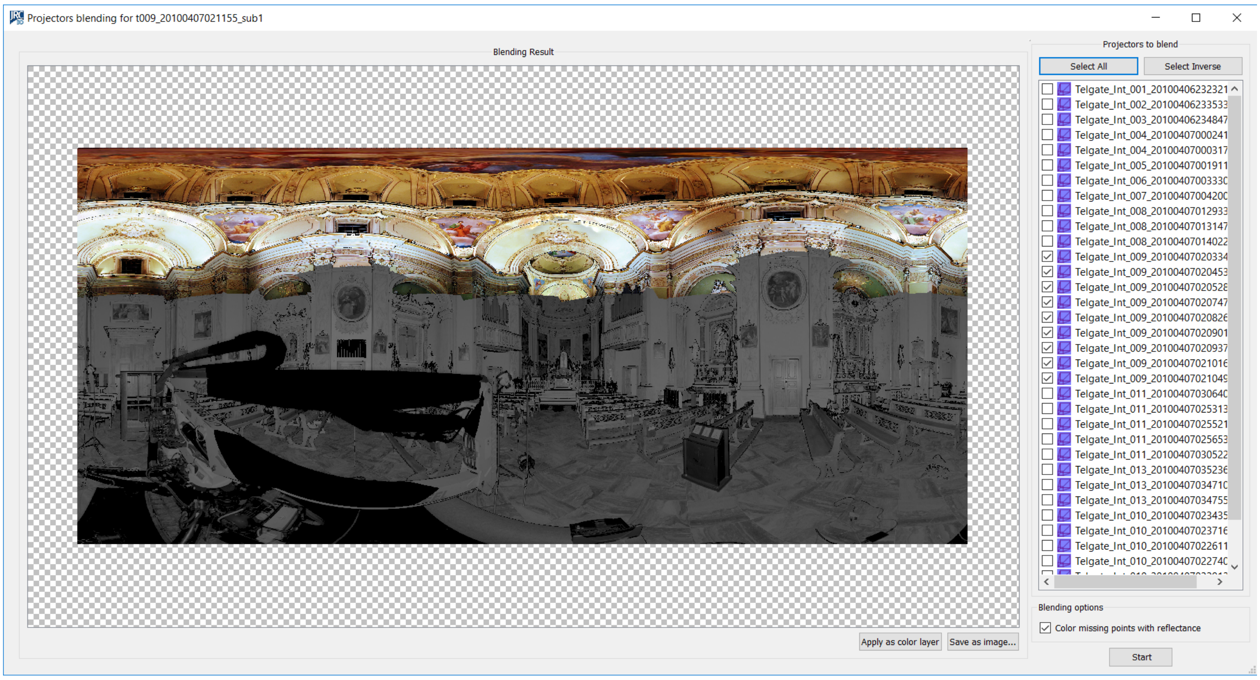
This command allows then to switch between scalar and color channels, customizing them.





Color clouds with photos

Projectors blending dialog



This dialog allows you to create a *color layer* for a *structured point cloud* by blending the images from selected projectors.
You can load all the *projectors* available in the current project. The color layer is computed by projecting on the point cloud and blending all the images coming from the projectors. Different blending algorithms are applied depending on the selected options.

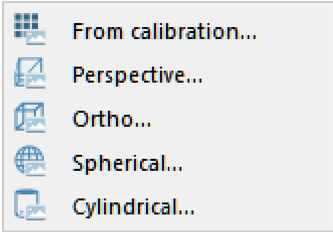
Blending Result: Shows the image resulting from the blending process.
Projectors to blend: Select the projectors to be blended by checking the items in the list. You must ensure that the projectors are properly calibrated for the selected point cloud, otherwise the correctness of result will not be guaranteed.
Apply as color layer: Add the blending result to the structured point cloud as a new color layer. Upon clicking the button you will be prompted for the name of the layer.
Save as image: Save the current blending result as an image. The image can be loaded at a later stage as a color layer (see Colors Mapping for further information on this subject).

If the "colors missing points with reflectance" option is checked, points which cannot be colored using projectors are colored with their reflectance value. If this opiton is left unchecked, the missing points are blacked out.

See also Texture mapping dialog and Camera calibration dialog.

Create Projector

This functions creates a projector.
Five different modes are available:



- **From calibration:** creates a projector (perspective or spherical) by importing a reconstructor camera calibration file (.cal);
- **Perspective:** creates a perspective projector;
- **Orthographic:** creates an orthographic projector;
- **Spherical:** creates a spherical projector;
- **Cylindrical:** creates a cylindrical projector.

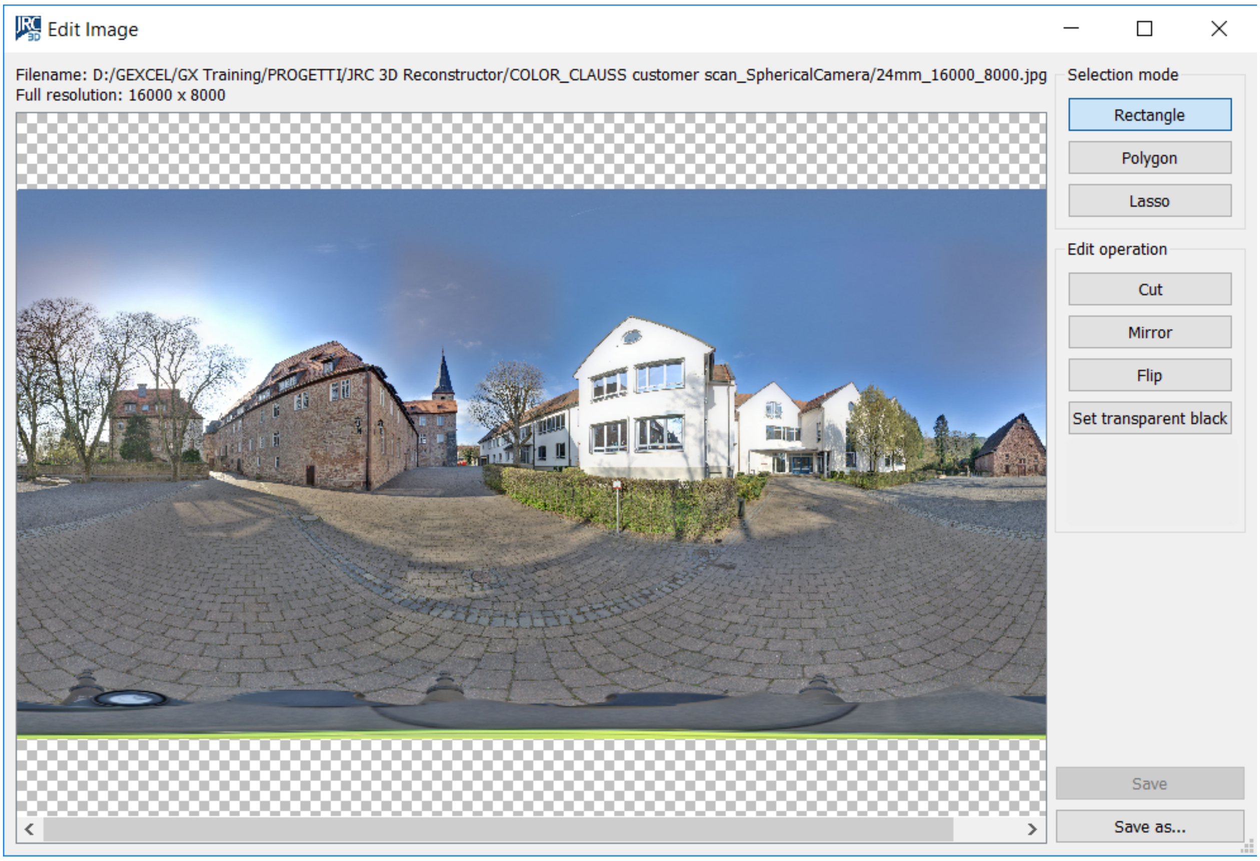
Note that a valid image is needed in order to create a projector.

See also Cameras, and Camera Calibration on a Grid.

Edit (projector) image dialog

This dialog is available only for projectors. Here is possible to edit the transparent parts of the image in order to optimize the blending of projectors.

The editing dialog of an image of a projector is accessible through the contextual menu of the loaded image (*Edit projector image*).



Data kindly provided by CLAUSS

Zoom in/out the grid with Alt+mouse wheel.

Selection mode

- **Rectangle**: press Left Mouse Button (LMB) for the first point and keep it pressed while dragging the mouse to the second point, then release the LMB
- **Polygon**: click LMB for each point of the polygon. Close the polygon by double clicking the LMB
- **Lasso**: press Left Mouse Button (LMB) to start the polyline and drag the mouse to draw, then release the LMB to close the polyline

Edit operation

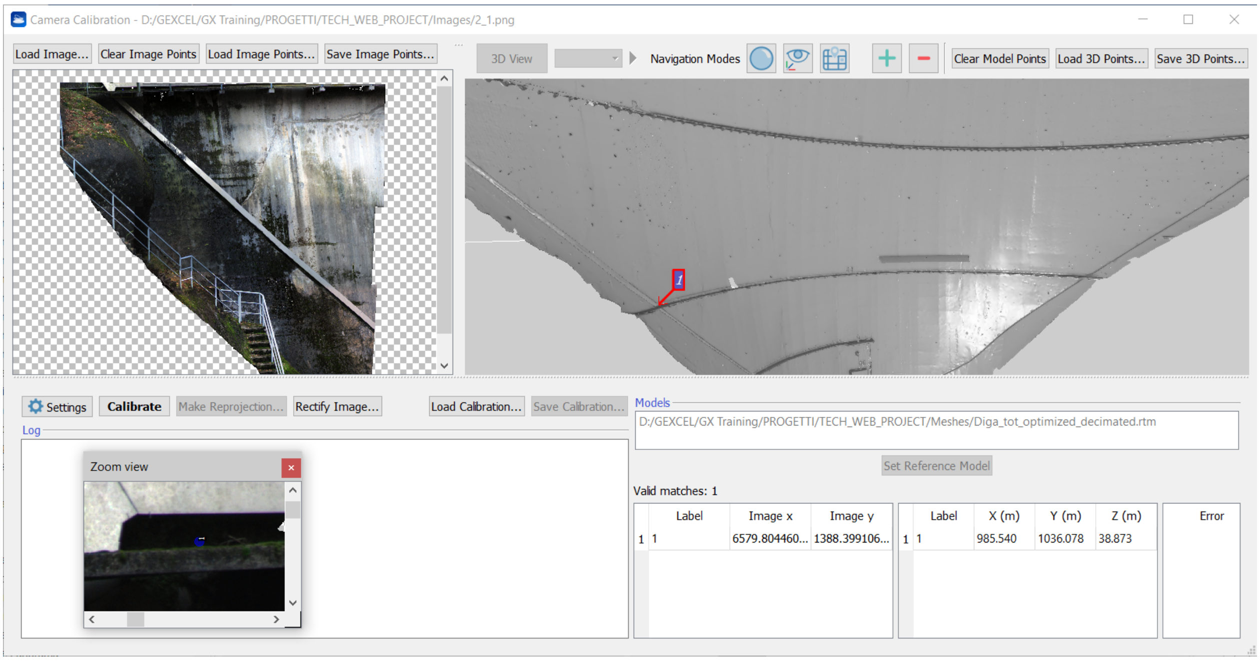
- **Cut:** set transparent the selected region by zeroing the alpha channel
- **Mirror**: inverts the image along the width
- **Flip**: inverts the image along the height
- **Set transparent black**: all pixels that are black, i.e. (0,0,0), are set to transparent by zeroing their alpha channel

Save is enabled only if a modification as been done and it's necessary otherwise the edits will be lost.
Save as allows to save a copy of the image to another file.

Camera Calibration on a model

The objective of camera calibration is the registration of photos (perspective or spherical images) with the 3D data.
This process creates photo-realistic texture maps for the 3D model.

When you access the command a *Camera calibration dialog* will appear:



Load a photo to calibrate with ***Load image***, and at least a grid point cloud with ***Load 3D grid***. If

necessary, change the reference grid from the loaded ones, the image-grid point pairs are stored for each grid, and are accumulated.
A resizable zoom window appears for both where an independent zoom factor (Alt-mouse wheel) can be set from the main window, just center the zoom area by moving the mouse on the main window while pressing [Ctrl] key. To place a marker, double click the left mouse button (LMB) either on the main window or in the zoom window. To move a marker, click on it with the LMB and keep it pressed while dragging the marker. To remove the marker, right click the mouse (RMB) on the marker.

In general, to interact with the images:

- *Zoom in/out*: mouse wheel
- *Fit the image in the window*: Alt + Middle Mouse Button (MMB)
- *Reset to original size*: Alt + Right Mouse Button (RMB)
- *Translate the image*: Shift + LMB drag
- *Space bar*: rotate image 90 degrees
- *Show/hide zoom window*: Z key
- *Move zoom region inside global view*: [Ctrl + LMB]

The list of points is shown in the table, where every cell is manually editable to force the values. It is possible to import a list of points from a file by pressing ***Load points***.

For the photo, the format must be
```
Image_x1 Image_y1 <Point1 Name>
Image_x2 Image_y2 <Point2 Name>
...
```

For the grid the format must be
```
X1 Y1 Z1 Image_x1 Image_y1 <Point1 Name>
X2 Y2 Z2 Image_x2 Image_y2 <Point2 Name>
...
```

Try to change the color type of the grid on top of the window to improve the contrast of features. Press the right arrow to show a menu of available commands for the current color type for better adjustment.

Choose the right calibration *Algorithm*, depending on your kind of photos:

- *Perspective* for perspective images
- *Spherical Camera* for spherical images
- *Fisheye* for photo acquired by fisheye lens

After at least 11 marker pairs (for perspective images and at least 4 for spherical images) are found between the photo and the grid, press ***Calibrate***. The result is shown in the Log panel with the computed field of view of the photo and relative re-projection error. If successful, the calibration and the point pairs can be saved to file by pressing ***Save calibration***. A *.cal* file is created, in XML format, storing the calibration using calibration parameters. As a test of the calibration, press ***Make Reprojection***, to save an image of the photo re-projected on the grid.

Point Guessing System

When you select at least 11 correspondences for perspective images (and 4 correspondences for spherical images), the point guessing system (PGS) will perform a background calibration and will automatically try to estimate the position on the image of additional points that you place on the grid.

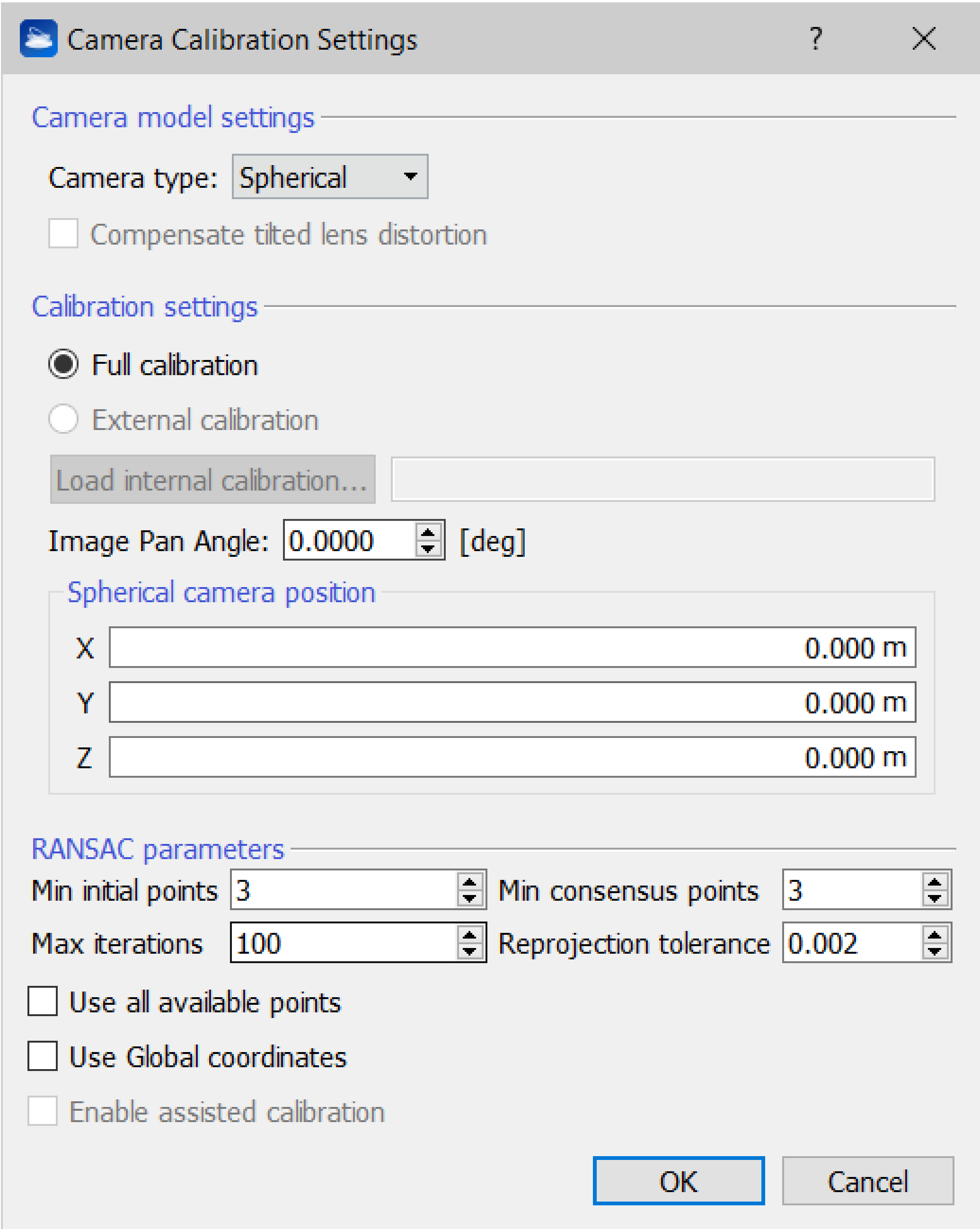
The procedure is the following:

- Select a point on the grid (right image).
- The PGS will place the guessed point on the image (on the left). The zoom box will center on the newly placed point.
- You have to correct the position of the placed point if necessary.
- If the PGS cannot place the point you have to place it manually.

Note: You can also select a point on the image and the PGS will try to place its correspondent on the grid, but this procedure is not recommended since it is slower and less accurate.

Camera Calibration Settings

By clicking on Settings, a dialog including all the camera model/calibration settings and RANSAC parameters is opened:



Camera model settings

Here you can select the kind of camera used to take pictures: Perspective, Spherical or Fisheye. For Perspective and Fisheye cameras it is possible compensate the tilted lens distorsion by flaggin the relative item.

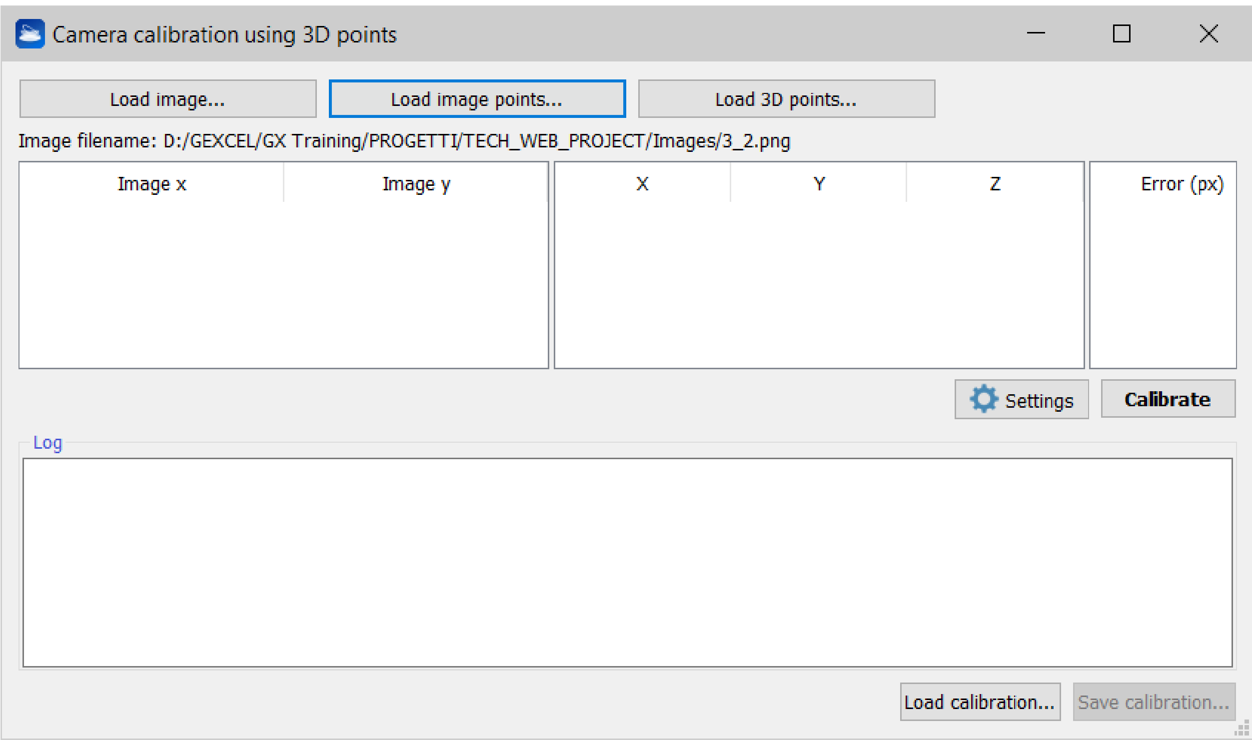
Calibration settings
Here it is possible to choose if running a Full calibration or only an External calibration, loading the .cal file of an Internal calibration already done.
If the image is rotate respect to the reference axis of the point cloud with a known value, you can set it in the *Image Pan angle [deg]* parameter. In the same way, if the relative position between the center of the point cloud and the center of a spherical camera is known, you can fill the X, Y, Z components of the rototranslation in the Spherical camera position cells.

RANSAC parameters
The RANSAC algorithm selects randomly the minimum number of points required to determine the model parameters (in this case the camera calibration), solving for the parameters of the model. Then determines how many points from the set of all points fit with a predefined *Reprojection tolerance*. If the fraction of the number of points over the total number points in the set exceeds a predefined threshold, re-estimates the model parameters using all the identified inliers and terminates. Otherwise, repeat the previous steps for*Max iteration* times.

See also the Point clouds coloring and textured meshes videotutorial.

Camera Calibration Using 3D Points

The objective of camera calibration is the registration of photos (perspective or spherical images) with the 3D data.
This process creates photo-realistic texture maps for the 3D model.

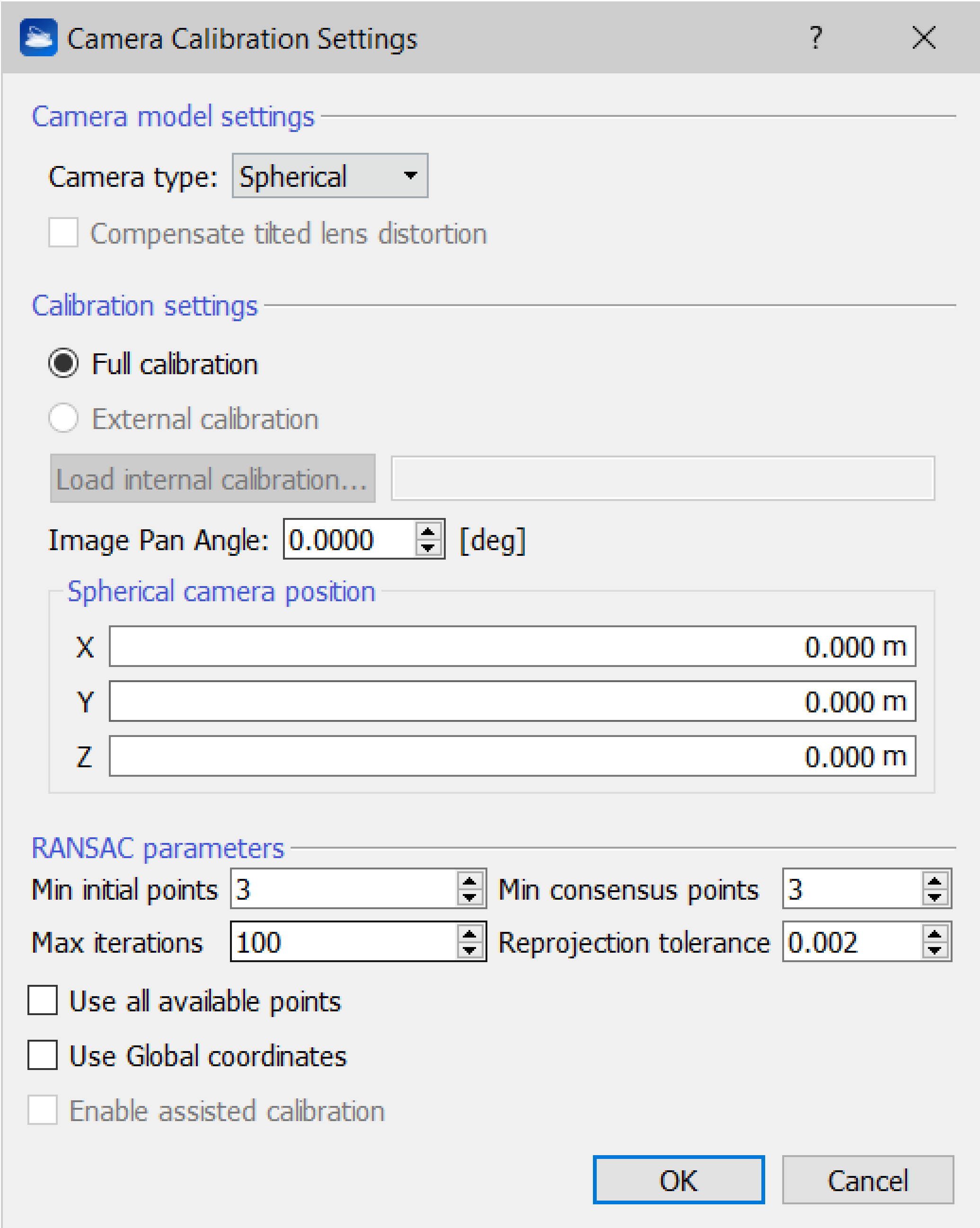


This dialog allows the calibration of a perspective image using a list of matches between camera points and 3D world coordinates points.

As first step, the image to be calibrated must be selected by using the ***Load Image*** button. Subsequently image points and 3D points must be loaded using the ***Load Image Points*** and ***Load 3D Points*** buttons respectively.
Then click on ***Calibrate*** to compute the calibration process. The result is shown in the Log panel with the computed field of view of the photo and relative re-projection error. If successful, the calibration and the point pairs can be saved to file by pressing ***Save calibration***. A *.cal* file is created, in XML format, storing the calibration using calibration parameters.

Camera Calibration Settings

By clicking on Settings, a dialog including all the camera model/calibration settings and RANSAC parameters is opened:

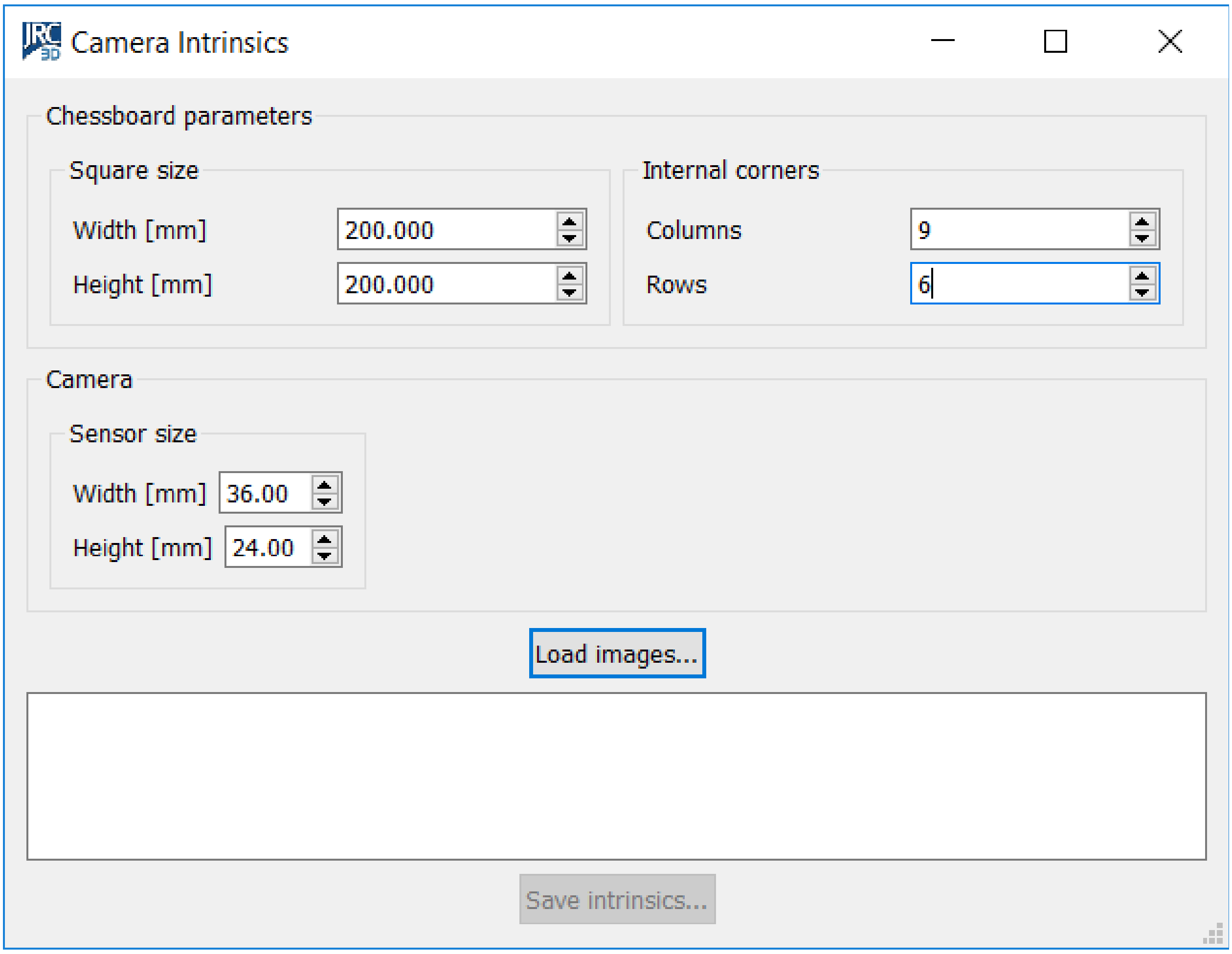


Please refer to the camera calibration page for details about the calibration options.

Calibrate Camera Intrinsics

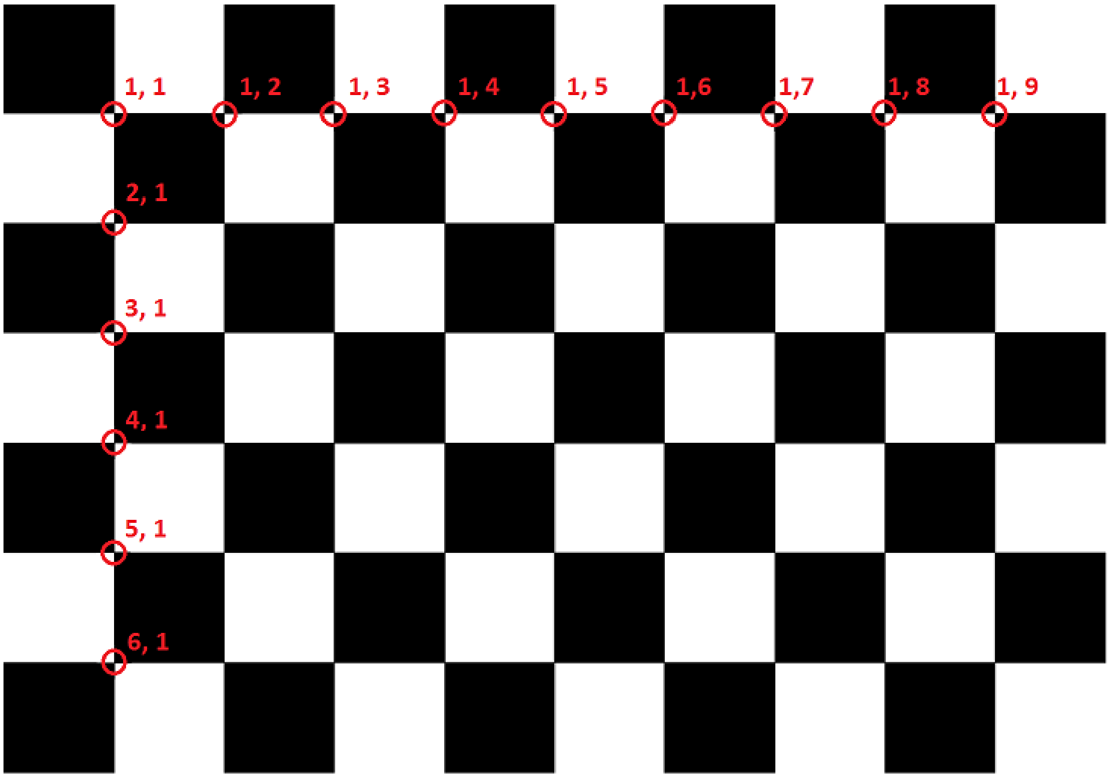
If you're using an external perspective camera (not connected to the laser scanner system) to calibrate high definition photos on grid point clouds, it could be necessary to create a specific calibration of the camera, save it and use it for future purposes.

Here is showed the procedure to calibrate perspective cameras intrinsic (or internal) parameters using multiple checkerboard images as calibration samples.



1. Print the calibration frame
2. Fixed it on a rigid support
3. Measure the high and length of the square
4. Select the camera and the lens you want to calibrate
5. Acquire at least 10 images (to have a robust estimate for the calibration parameters) of the frame changing the camera position, paying attention to include all the frame squares in the picture, in order to reduce the image without the frame and to avoid glare effects
6. Create a new Reconstructor® project then enter in the command *Tools → Color tools → Calibrate Camera Intrinsic*
7. Set the square size and the number of internal corners and load the frames images

Chessboard square size: the width and height in millimeters of a single chessboard square;
Chessboard internal corners: the number of corners of the chessboard, the border corners should not be taken into account;
Camera sensor size: the width and height in millimeters of the camera sensor. Refer to the camera technical documentations to obtain this data.



In this sample image, a chessboard with 9 by 6 internal corners is shown. Please note that, in order for the algorithm to work properly, the number of row and column corners must be different.

1. Calibration process starts automatically
2. At the end of the calibration process, check the mean re-projection error to verify the process quality
3. Press “Save intrinsic” to store the calibration file (Internal-Calibration.cal) usable for all the other pictures

Once the calibration is complete, a calibration file can be saved. Such file can be used as a starting point for camera parameters while using the camera calibration function. Inside the Camera calibration on a grid dialog, select *External* and load *Internal calibration file* just created.

- You can download here the calibration chessboard to calibrate your camera.
- After selecting the camera and fixing the focal length, the internal calibration can be done once for several acquisition jobs
- After an intense usage of the camera, we suggest you to verify the calibration parameters

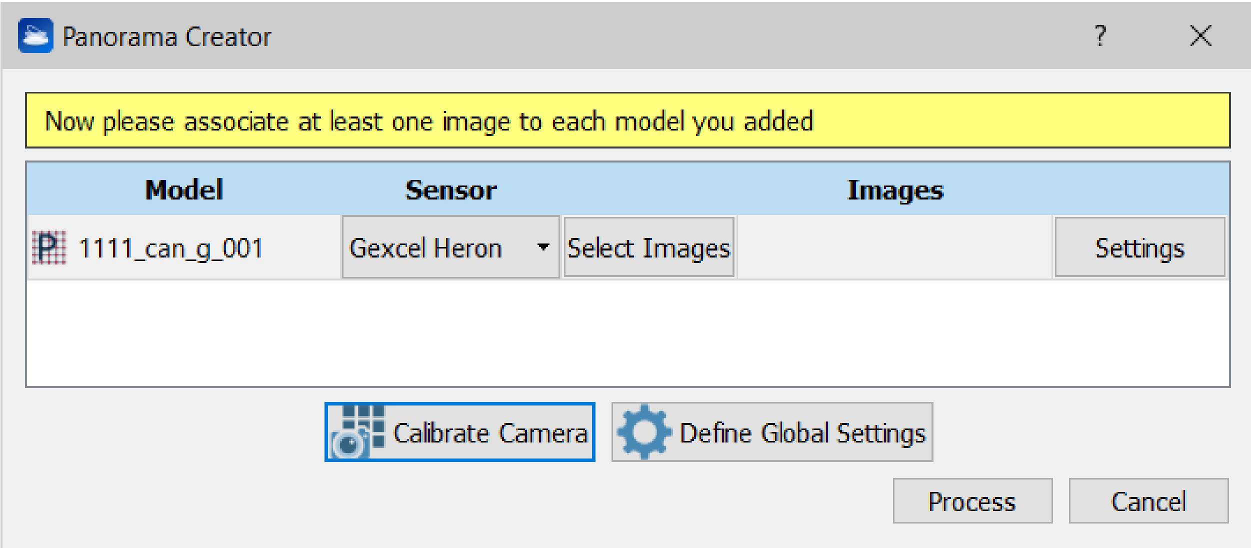
Create panorama

Create Panorama tool creates and associates a panoramic spherical projector to a point cloud (both structured and unstructured), by calibrating a series of perspective images on the point cloud.

Some requirements are due:

- The images must be taken from the same point of view
- only a regular rotation of camera is permitted (i.e. an image every 30° during camera rotation), but a complete 360° rotation is not mandatory.

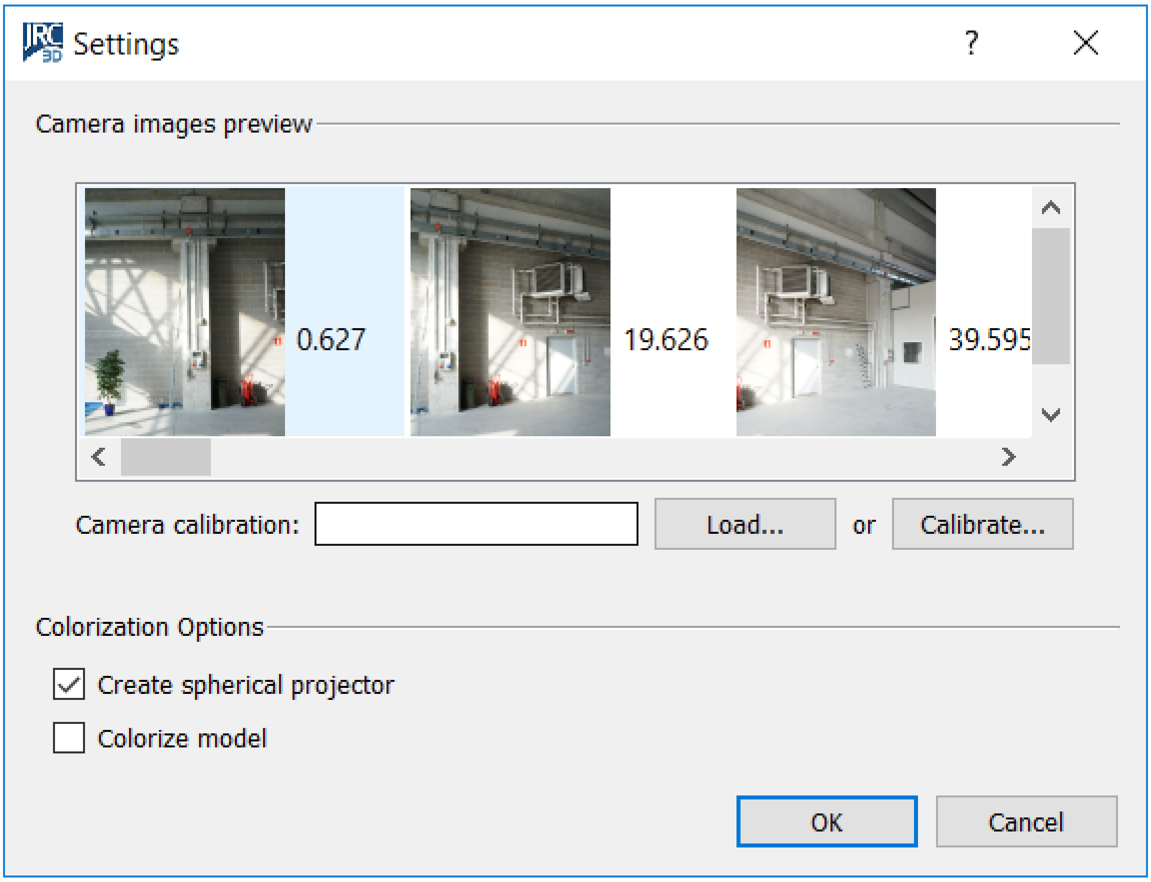
1. Start by Dragging & Dropping at least one model from the tree view in the Project Window
2. If the sensor which acquired the scan is not defined (ensor Unknown), select the proper one in the drop down menu
3. Associate at least one image to each model by selecting it



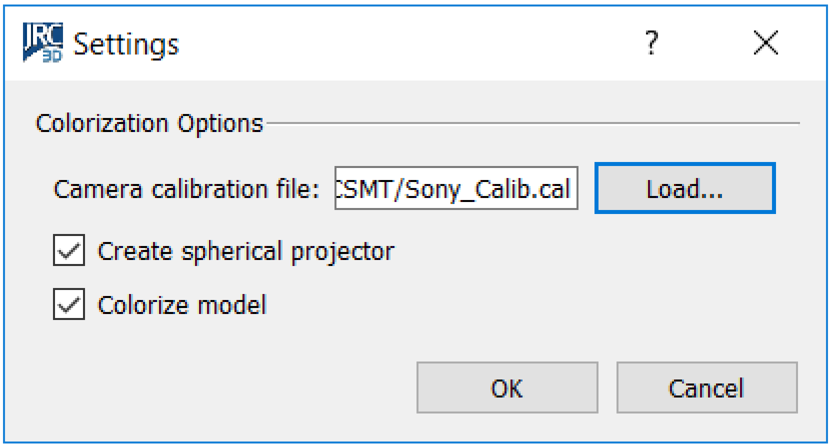
4. Calibrate Camera: select an image (among the imported ones) and proceed with the camera calibration (as explained in Camera calibration on a grid).

Settings

Calibration settings can be singularly adjusted for each model



or globally defined for every model (*Define Global Settings*) in a similar way.



UAV and Imaging

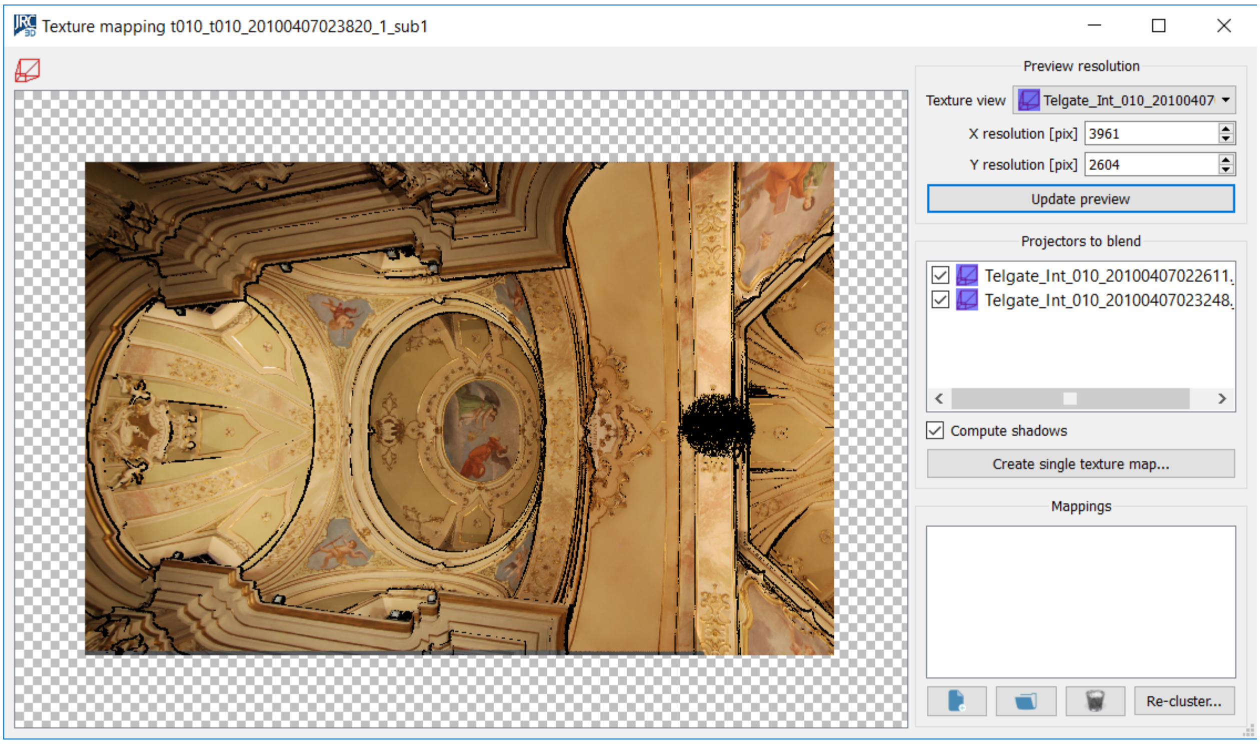

These tool is used to Import GeoTIFF.

Import Georeferenced TIFF


This function imports a geo-referenced TIFF as an orthographic camera.
The image to be imported must be accompained by a Tiff World File (TFW) indicating the coordinates of the image.

Texture mapping

The command *Create texture map* allows you to create *textures* for your mesh model.



Preview resolution

Firstly select a camera to specify a viewpoint from which the texture should be created and parameterized. The resolution of the camera can be changed here.

Update preview: creates a preview of the texture from the selected view point and projection. This is a simplified image and does not represent the full quality of the final texture

Projectors to blend

You can load simultaneously more *projectors* that project photographic color on the same mesh.

⚠ *The maximum number of simultaneous projectors supported depends on your graphic card. Check at Help > Hardware info | GPU and graphic functionalities what is its value.*

Successively, you can create a *texture*.

A **texture** is a particular image that, mapped on your mesh, gives to it the color. The texture is calculated by blending together all the color coming from the projectors.

Create single texture map: create the texture by blending the checked projectors. An image of png format is created, plus a 2f file that stores the computed texture coordinates for the mesh vertexes, which is listed in the Mappings. Check if shadows must be computed.

Mappings

- **Remove**: remove the selected mapping from the list

- **Remove all**: clears the list

Re-cluster: create a new triangle meshes model where the listed mappings are used to re-arrange the triangles in a new set of submeshes that map the desired textures. Each triangle is mapped with the best texture quality available by computing the best texture projection among all. The model is automatically added to the project.

In the contextual menu of meshes the command **Sample texture as vertexes** is available.
It assigns to the vertices of the mesh a color obtained from the corresponding pixel of the texture.

Results & Analyses

This menu contains many functions to extract results from the project's data and models. These functions are organized in the following submenus:

- Measures & notes
- Elevations & plans
- Cross sections
- Areas & Volumes
- Inspection
- Drawing tools
- Video record
- Save snapshot
- Manage Units of Measure

Measures & Notes

In Reconstructor® some basic measure feature are available:

Readout window
The readout window allows you to see the 3D coordinates of the points you are hovering the mouse on in the 3D rendering window.

Annotation
This function allows you to create annotation of 3D points of models in your project.

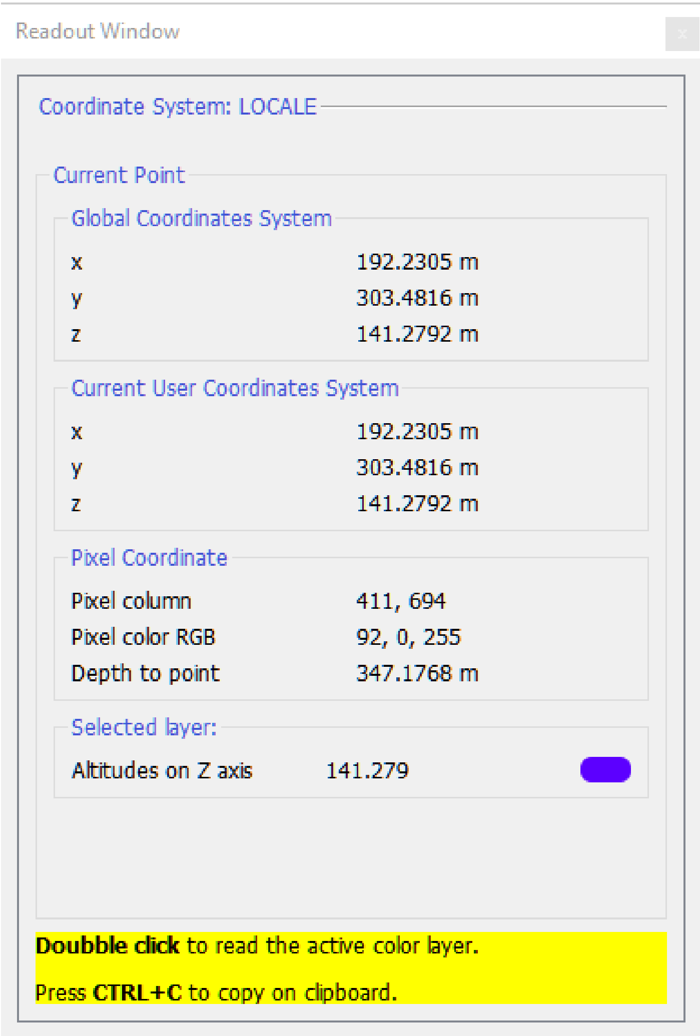
Distance
This button allows you to measure a distance between any two points in the 3D scene

Angle Tool
This button enables you to to measure an angle identified by three points directly picked in the 3D window.

Open Angle Tool
This button enables you to measure an angle between different geometric shapes.

See also Areas & Volumes to learn about areas and volume measurements in Reconstructor®.

Readout window



This window shows in real time the current coordinates and color of the models you see in 3D rendering window below the mouse pointer.
To obtain the accurate coordinates, double left mouse click and stay in position.

The coordinates of the corresponding 3D point are expressed in the Global Coordinate System (MAIN UCS) and in the Current UCS.

- *Pixel column, row:* refers to the mouse position inside the 3D window, in pixels (0, 0 is the top-left corner)
- *Pixel color RGB*: the three components of the point's *RGB* are shown
- *Depth to point*: distance between selected point and current view point is displayed

Values of the current layer can also be displayed after the double click.

You can copy all the values to the clipboard by *Ctrl+c*.

From the *Change units of measure* button the Select Units of Measure window will appear, permitting the change of displayed units of measures (SI, imperial or U.S. customary units).

The readout window appears docked by default in the right area of the GUI.

Example

```
Current Coordinate System: LOCALE

Global coordinates system:
x: -0.12699 m
```

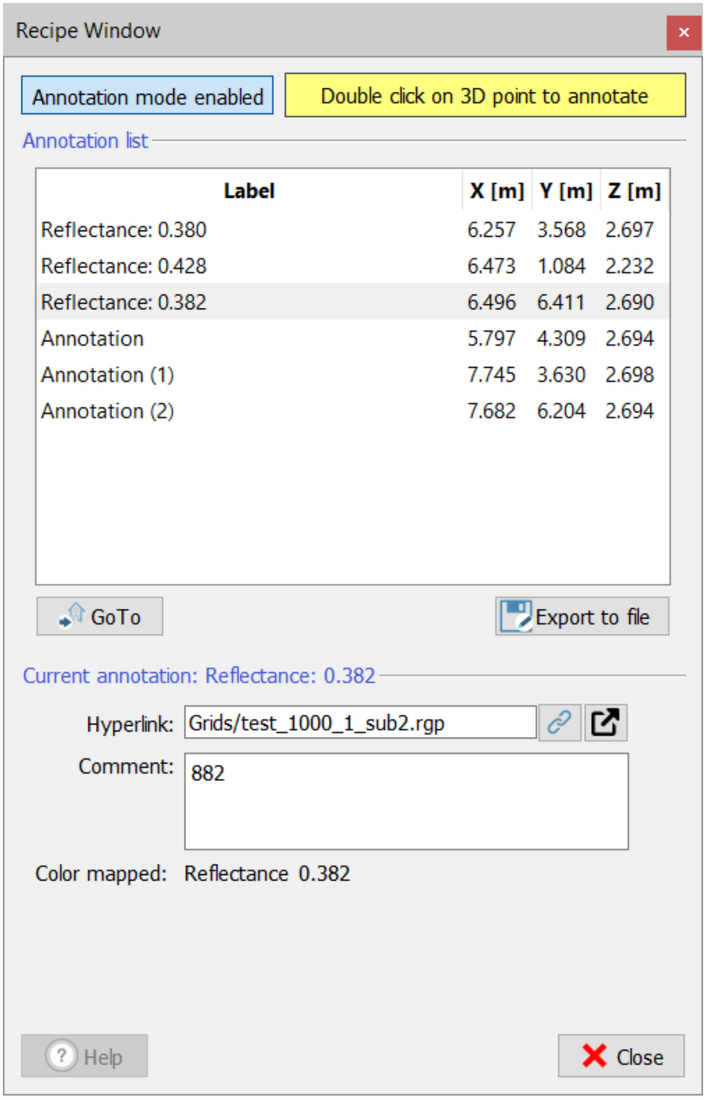
```
y: -0.04696 m
z: -18.04474 m

Current User Coordinates System:
x: -0.12699 m
y: -0.04696 m
z: -18.04474 m

Pixel Coordinates:
Pixel column,row:    511, 448
Pixel color RGB: 224, 255, 0
Depth to point:  1.50784 m

Altitudes on Z axis: -18.0447
```

Annotation



This function allows you to create Annotations of 3D points of models in your project.

After activating this button, double-click with the left mouse button on a valid 3D point to place an annotation. The annotation gets displayed in 3D with a label and it also appears in the "Annotations" group of items in the Project Window.

In the Annotation list all the picked annotations are listed, including their Labels and X, Y, Z coordinates in the current UCS.

By clicking on *GoTo* button the 3D view is focused on the annotation point.

You can edit the annotation's comment and set an associated hyperlink (by default it is the model on which you pick the point).

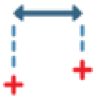
From the Property browser, you can change the color ID, edit the displayed comment, define a hyperlink for an annotation and choose whether to draw the annotation or not.

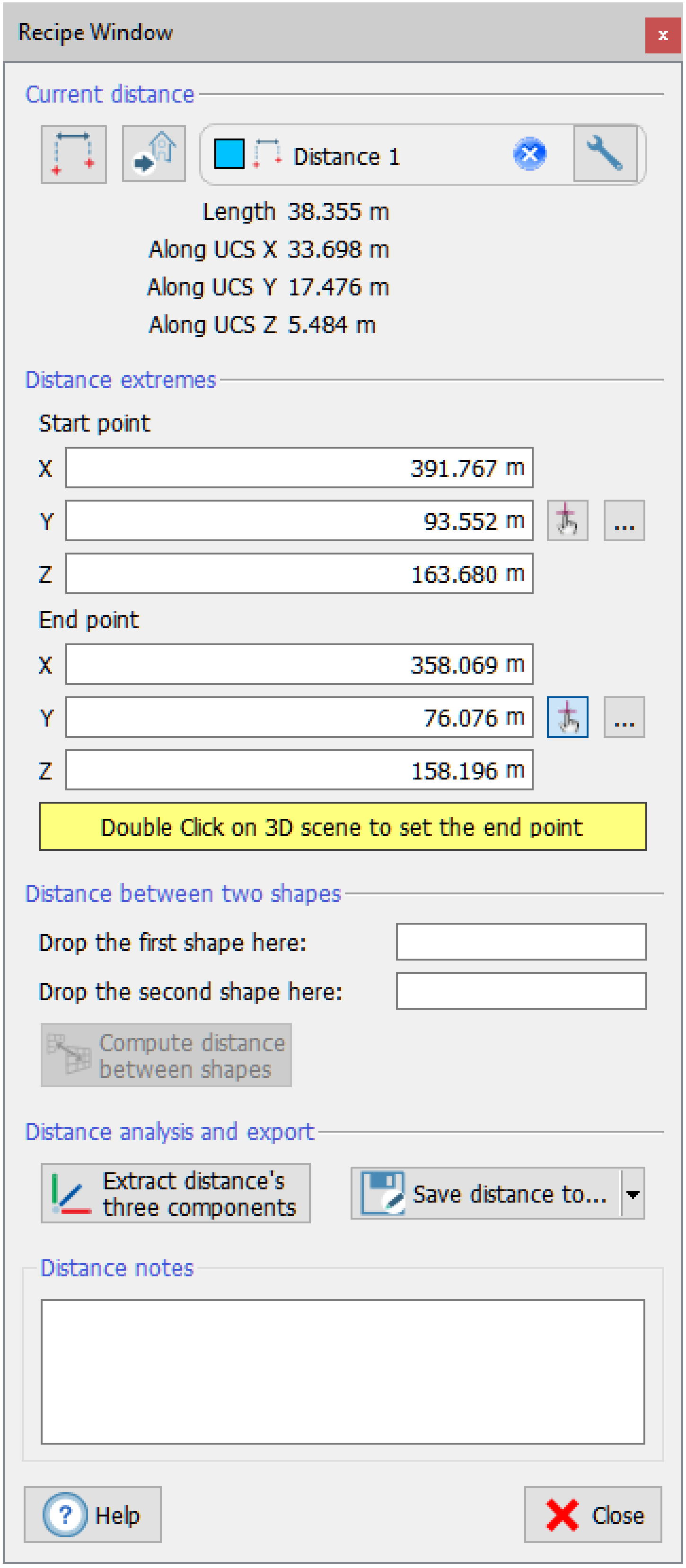
Export to file

Using this function, you can Export annotations and import them in spreadsheets or text editor third party software for further processing.

Annotations can be converted to *Points* or *Targets* items through the contextual menu, after selecting them in the Project Window. It is also possible to do the reverse.

Distance





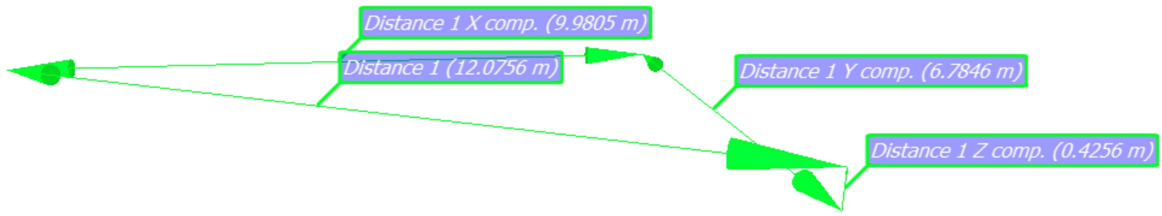
This function allows you to measure a distance between any two points in the 3D scene or to edit a previous one.

Start by dragging and dropping in the editor window the distance you want to edit. If no distance is dropped, a new distance will be created when you click on .

To define the starting and ending points, just click on  and double click on any point in the 3D scene. Otherwise, select any point from the point list window by clicking on ... . It is also possible manually insert the coordinates of the points in the dedicated boxes. When you have defined the points, a segment (with arrows) will appear in the 3d view, as well as its length's value in the dialog.

By dragging&dropping two shapes in the apposite boxes in the dialog it is possible to compute the distance between them.
See also Distance between shapes.

By clicking on *Extract distance's three components* the components along X, Y and Z directions (of the current UCS) will displayed in the 3D view.



The *Save distance to...* command allows you to:

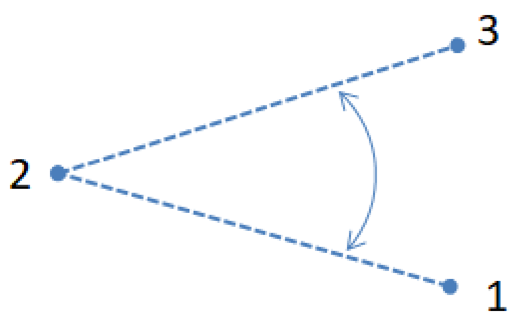
- save distance's extremes a s points in the project
- save the distance's information (below illustrated with an example), in .txt, .csv formats and as a clipboard.

```
Distance name: Distance 2
Length: 2.6562 m
Along UCS X: 0.0273 m
Along UCS Y: 0.0115 m
Along UCS Z: 2.6561 m
Start point coordinates: 0.4548, 0.6176, 5.4918 m
End point coordinates: 0.4821, 0.6061, 2.8357 m.
Annotation: Distance between windows
```

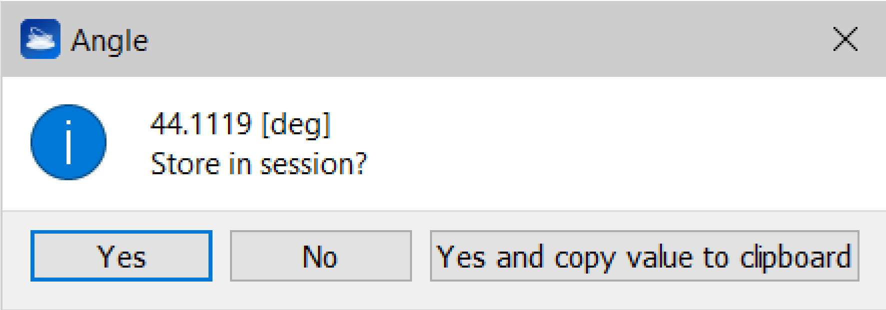
In the last box of the dialog, annotations (notes) can be associated to the distance.

Angle Tool


This function allows you to measure an angle identified by three points directly picked in the 3D window:

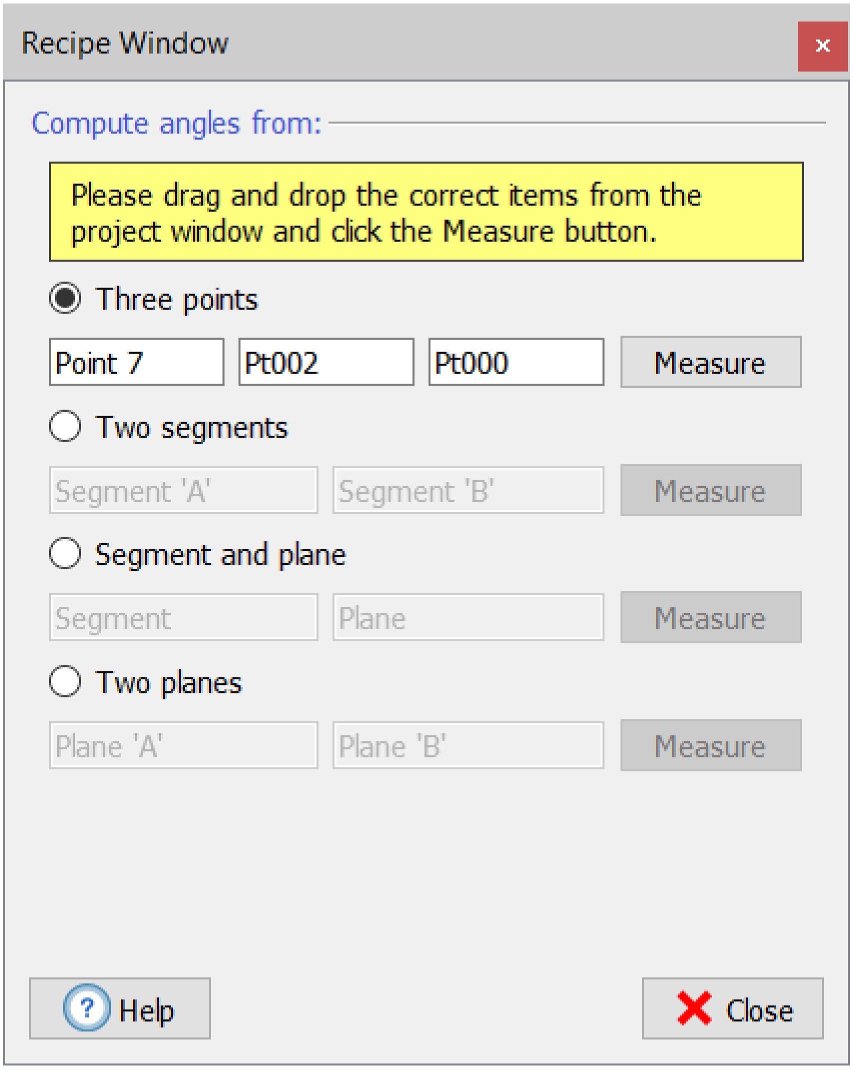


The result is an angle measure that you can store in your project session,



visualize in the 3D window, meanwhile a new *Angle annotation* will be inserted in the *Survey tools* list.

Open Angle Tool



This function allows you to measure an angle between three points, two segments, a segment and a plane and between two planes.

Drag and drop the objects mentioned above in the recipe window, then click on *Measure* button to take the measurement and automatically store it in the *Survey tools* list as a new Angle annotation including angle measure (e.g. 49.08 ° angle).

Elevations & Plans

This sub-menu contains all the commands and features to create precise, high-resolution elevations and plans of your models:

Create Orthophoto
Creates an Orthophoto and defines its position, size and field of view based on different parameters and methods.

Create X-Ray Orthophoto
Creates an X Ray Orthophoto from a set of selected point clouds. Main edges and borders are emphasized and displayed in the final Ortho view.

Orthophoto viewer
Opens an orthophoto in the viewer to measure distances, angles and areas.
From the viewer, the orthographic image can be exported in AutoCAD.

Create Camera
Creates four types of camera (perspective, ortho.

Place here survey point
Creates a perspective camera placed in the current view point.

Virtual scan
Computes a Virtual Scan of the models in the 3D scene from a selected camera. The virtual scan can be saved as a 2D image, or as a grid point cloud, and exported to AutoCAD as a scaled Orthophoto.

Create Orthophoto

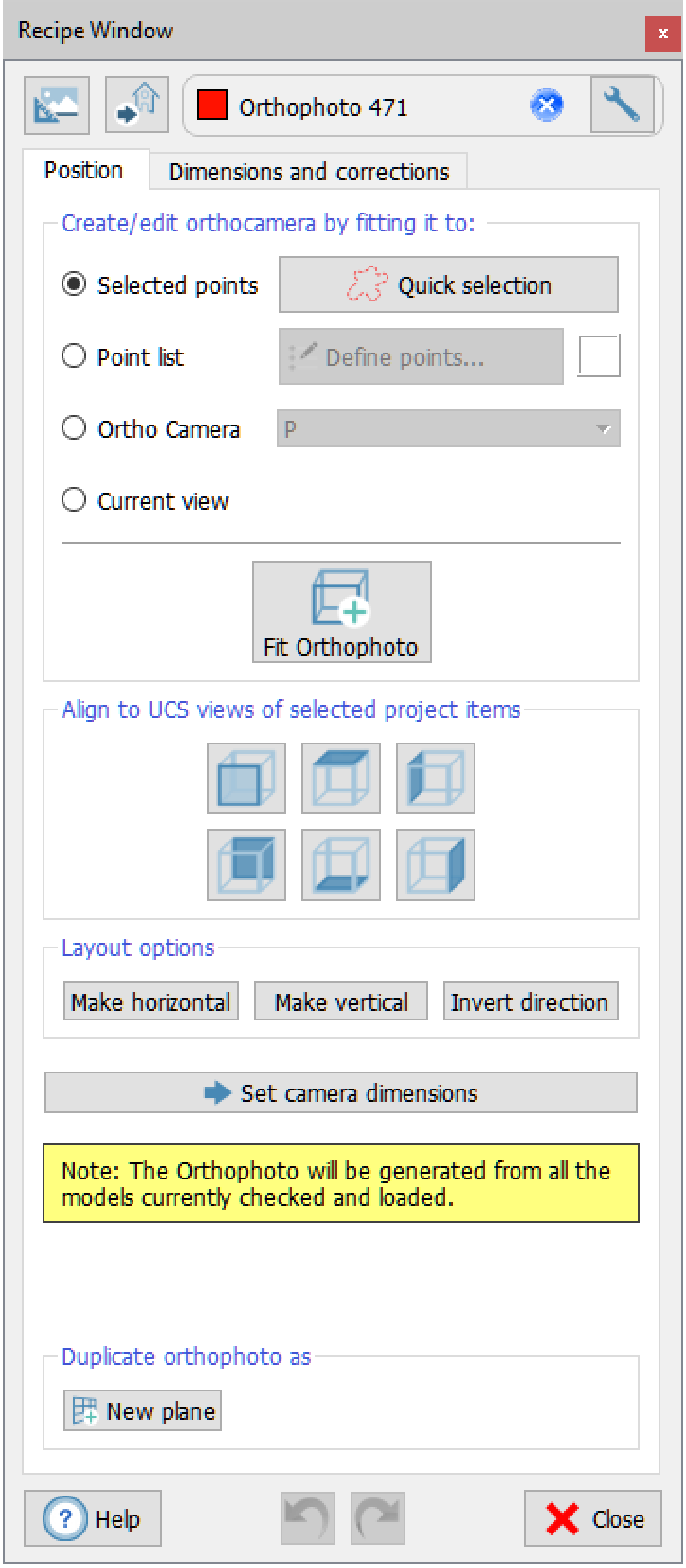
With this tool you can create an orthocamera from samples, points, axes, UCS directions, etc., edit it and extract an orthophoto.

An orthophoto is an orthorectified 2D image such the scale is uniform across the entire image.

The recipe window is splitted in two panels:

- Position
- Dimensions and corrections

Position



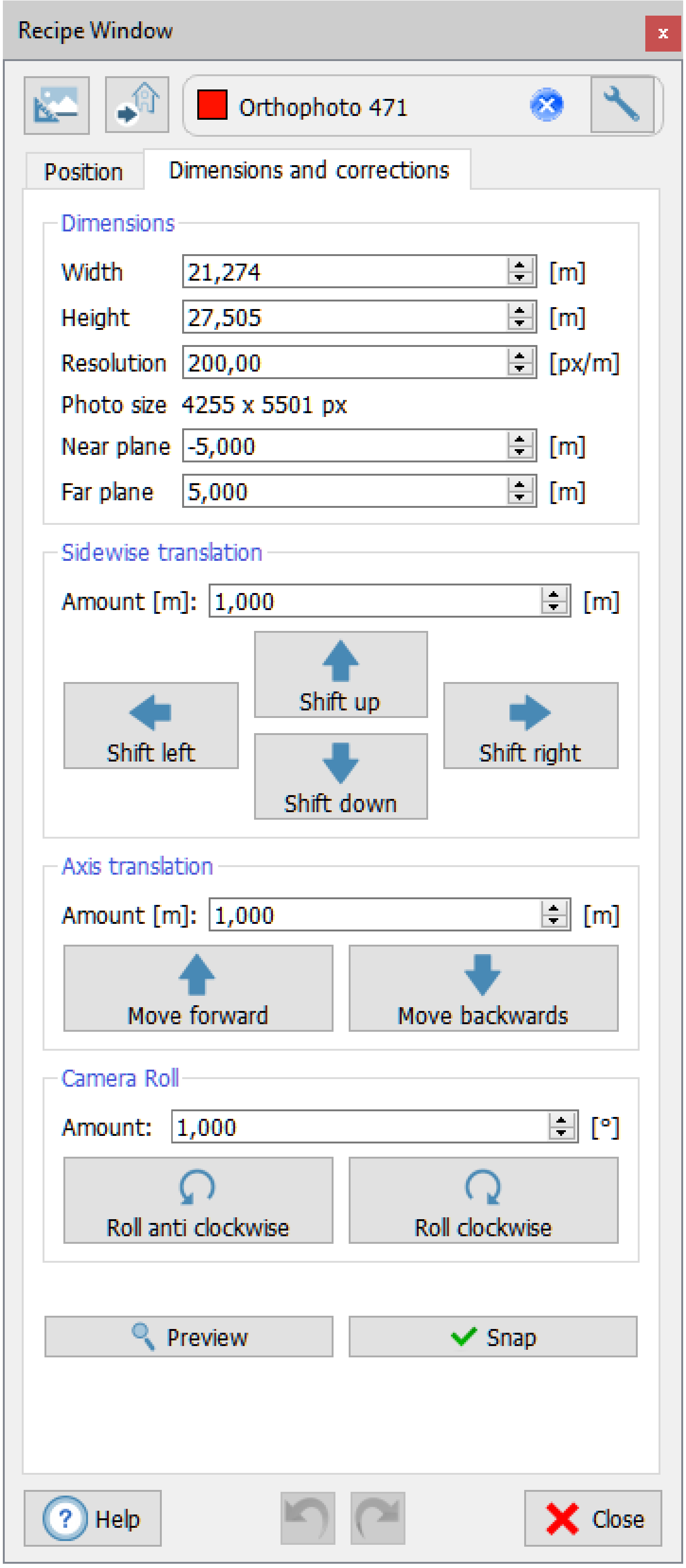
First of all press the *Create new*  button.

The panel of the dialog allows you to create an orthocamera from selected samples and a *Quick selection* (as in Selection tools) or from a Point list. By pressing *Fit Orthophoto* button an orthophoto is created in 3D window and in the project tree.
Then it's possible to edit the camera, aligning it to UCS views, making it horizontal or vertical,...

At the end of this process press *Set camera dimension* button to switch in the *Dimension and Corrections* panel.

The Orthophoto will be generated from all the visible models (loaded and flagged in the Project Window).

Dimensions and corrections



Here you can set the properties of the camera:

- dimensions
- resolution
- near and far plane distance.

The position of the camera can be changed through translations and rotations in the space, as you can directly see in the 3D window.

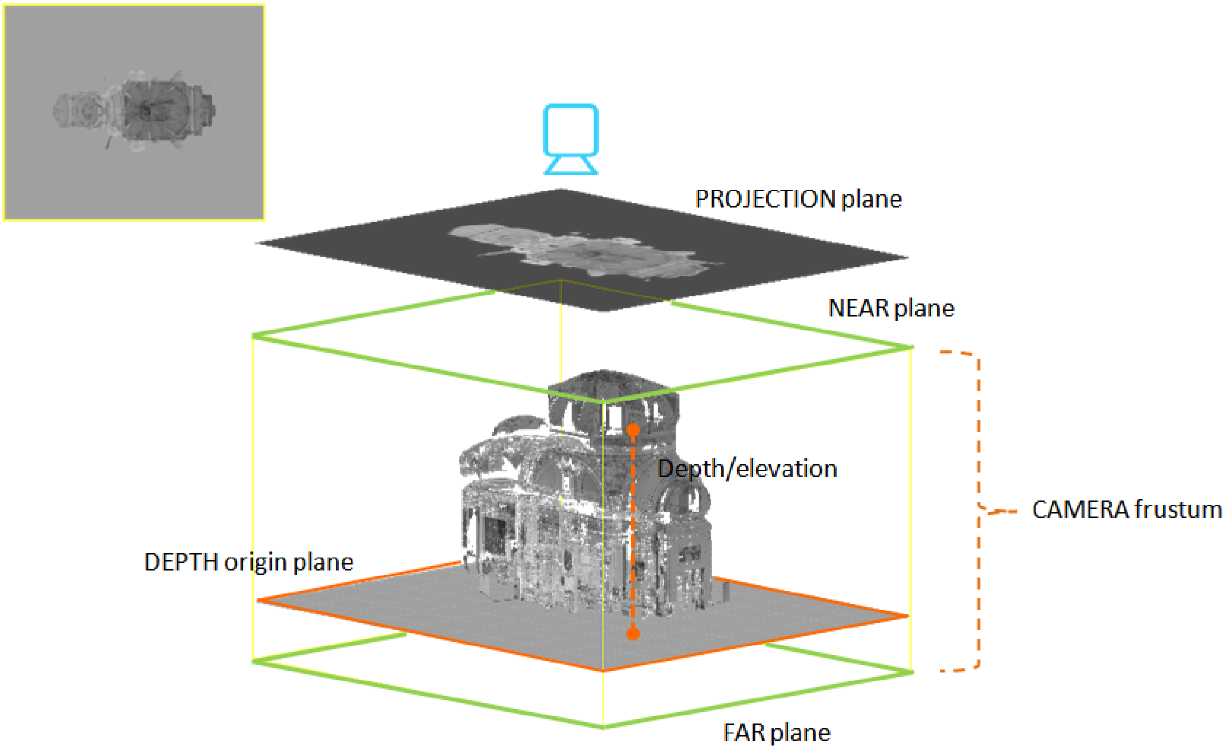
Depth

By selecting the *Depth* tool, the distance between a point on the orthophoto (1 point = 1 pixel) and a reference level (depth origin plane) is added in the properties of the orthophoto. The main purpose is to create a spot height map.

The difference between *Depth* and *Elevation* options lies in the sign; setting zero on the depth origin plane:

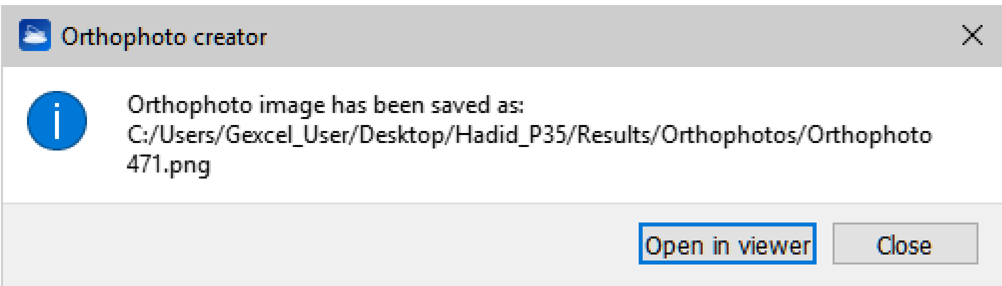
- in altitude mode, all values above the depth origin plane (e.g. between depth origin plane and projection plane) are positive and all values below are negative;
- in depth mode, all values below the depth origin plane are positive and values above are negative.

The image below can better show you the relationship between all the geometric components of the orthophoto structure.



The ***Preview*** button allows you to visualize in the 3D window a preview of the orthophoto, setting the same point of view and visualizing the loaded and checked 3D models. It helps you to find the right position of the camera, by showing you a preview of the final result.

At the end of this process press ***Snap*** button to create the orthophoto then open the result in the Orthophoto Viewer pressing the *Open in viewer* button.



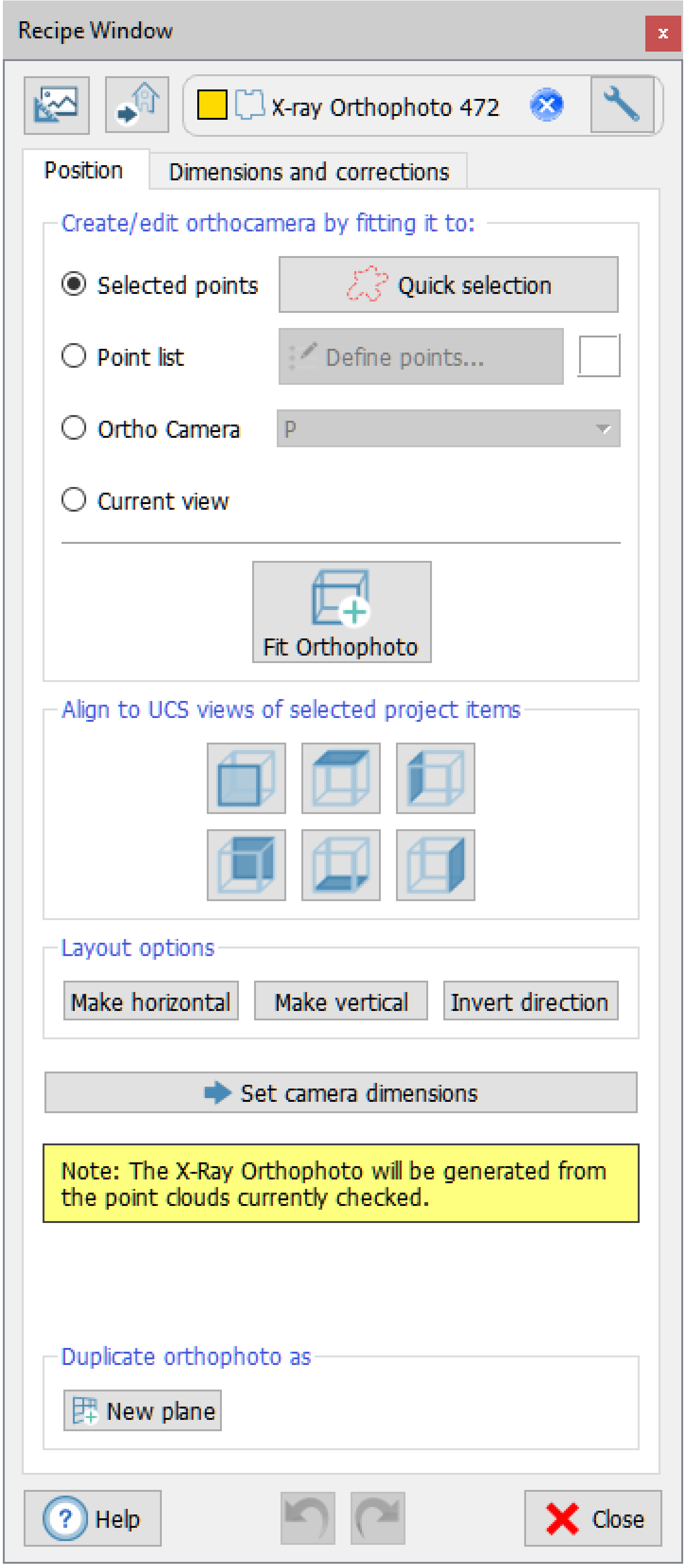
Create X-Ray Orthophoto

With this tool you can create an orthocamera from samples, points, axes, UCS directions, etc., edit it and extract an X-Ray Orthophoto.
X-Ray Orthophoto can enhance important features of a 3D object such as edges, corners, walls... It emphasize points aligned in one direction of view.
Only point clouds can be included in an X-Ray Orthophoto snap.

The recipe window is splitted in two panels:

- Position
- Dimensions and corrections

Position



First of all, press the *Create new*  button.

The panel of the dialog allows you to create an orthocamera from selected samples and a *Quick selection* (as in Selection tools) or from a Point list. By pressing *Fit Orthophoto* button an orthophoto is created in 3D window and in the project tree.

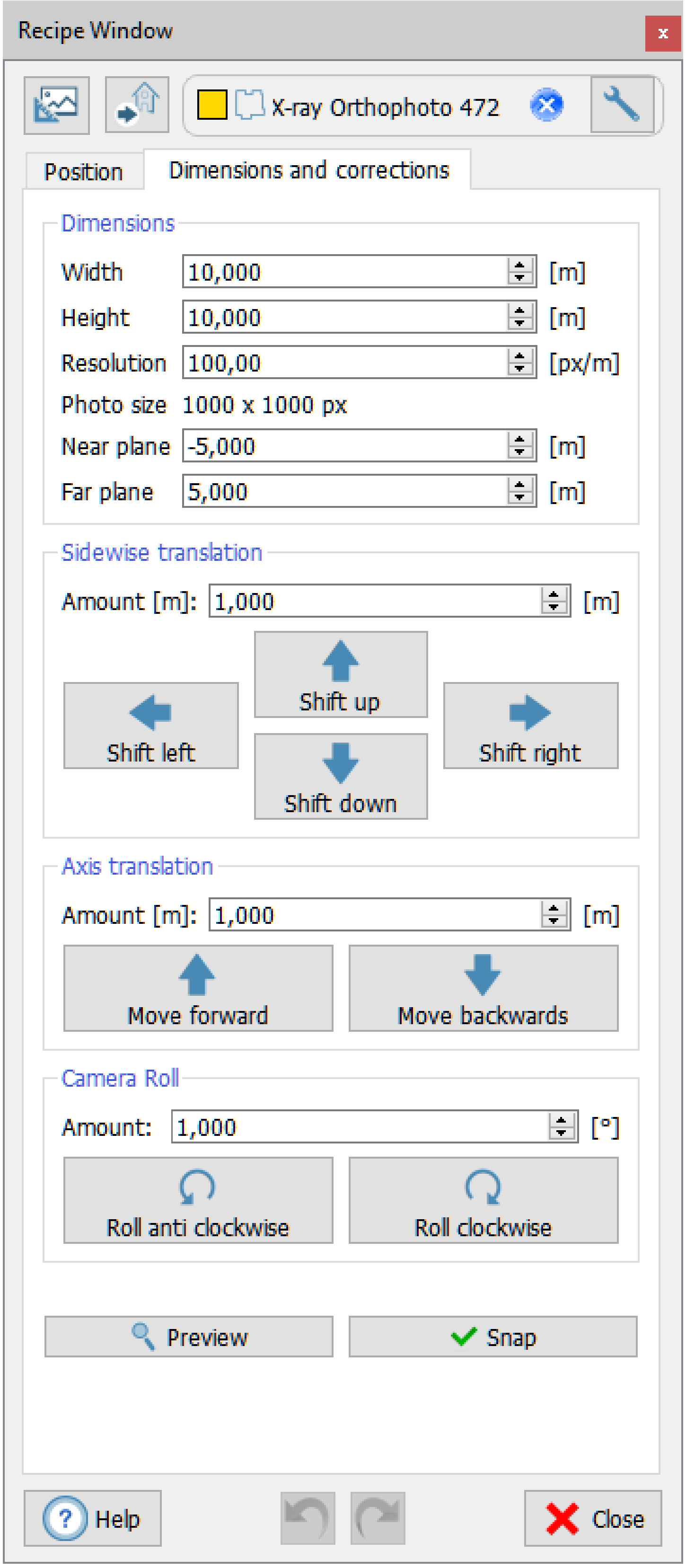
Then it's possible to edit the camera, aligning it to UCS views, making it horizontal or vertical...

At the end of this process press *Set camera dimension* button to switch in the *Dimension and Corrections* panel.

The X-Ray Orthophoto can be generated from:

- all the clouds in the project
- only the currently checked clouds
- only the currently loaded and checked clouds
- a custom selection of point clouds

Dimensions and corrections



Here you can set the properties of the camera:

- dimensions
- resolution
- near and far plane distance.

The position of the camera can be changed through translations and rotations in the space, as you can directly see in the 3D window.

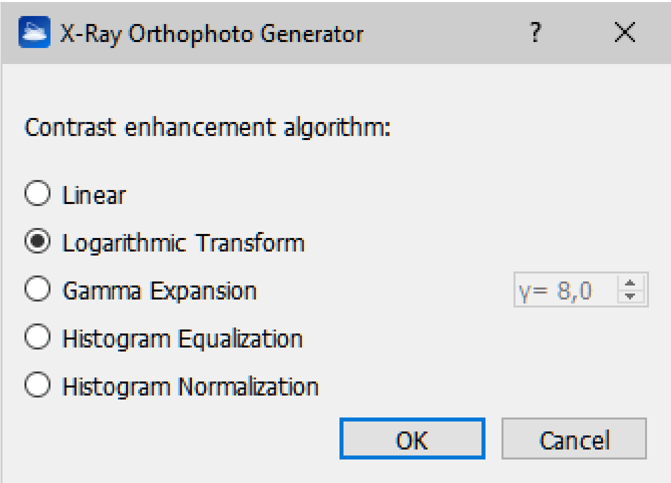
Depth

By selecting the *Depth* tool, the distance between a point on the orthophoto (1 point = 1 pixel) and a reference level (depth origin plane) is added in the properties of the orthophoto. The main purpose is to create a spot height map.

The difference between *Depth* and *Elevation* options lies in the sign; setting zero on the depth origin plane:

- in altitude mode, all values above the depth origin plane (e.g. between depth origin plane and projection plane) are positive and all values below are negative;
- in depth mode, all values below the depth origin plane are positive and values above are negative.

X-Ray Orthophoto settings

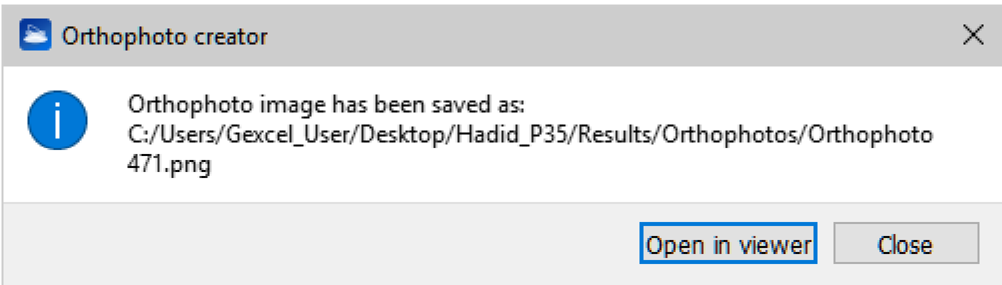


Select the desired contrast enhancement algorithm during the X-Ray Orthophoto generation.
For the majority of cases the default (Logarithmic Transform) works well, feel free to try others too, specific for your needs.

Press *OK* and wait a while.

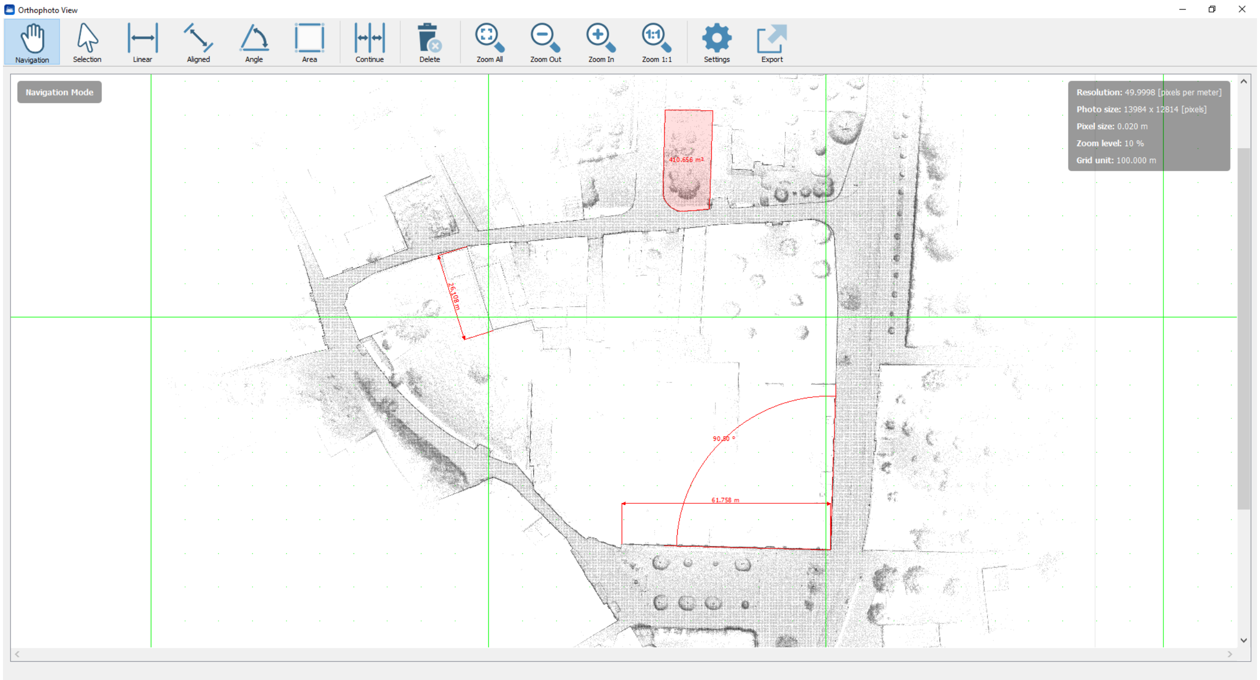
The ***Preview*** button allows you to visualize in the 3D window a preview of the orthophoto, setting the same point of view and visualizing the loaded and checked 3D models. It helps you to find the right position of the camera, by showing you a preview of the final result.

At the end of this process press ***Snap*** button to create the orthophoto then open the result in the Orthophoto Viewer pressing the *Open in viewer* button.



Orthophoto Viewer

To open a 2D viewer to visualize and dimension 2D maps created in Reconstructor (both x-ray orthophotos and standard orthophotos).



All the functionalities are below explained.

Navigation Mode

Navigation [*Alt+1*]
Pan the view by clicking the left mouse button and releasing it when the desired view is achieved.

Zoom out
To decrease the orthophoto zoom level.

Zoom in
To increase the orthophoto zoom level.

Zoom to Fit
To zoom to fit the entire orthophoto.

Zoom 1:1
To zoom to the orthophoto pixel size.

Selection [*Alt+2*]
Click on a item (e.g. a dimension) to add it to a multiple selection. Click on it once again to unselect it.

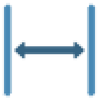
Click a point to start a selection rectangle, drag out a box then click to another point to complete the selection.
A different selection is due to a different direction by creating the rectangle:

- Left to Right: to select fully enclosed items
- Right to Left: to select partly enclosed items

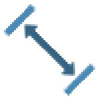
Press *Esc* to cancel the rectangle

Double click or press *Esc* to deselect all items.
Press *Del (Canc)* or the command *Delete* to delete the selection.

Measure Mode

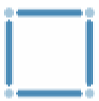


Linear [*Alt+3*]
To add a linear dimension, X or Y aligned.
Click to pick the measure starting point.
Move the cursor to seek the desired position of the dimension and then click when you found it.
Hold the *Alt* key to axis-align the measure during the point selection.



Aligned [*Alt+4*]
To add a linear dimension, aligned to the distance between the ending points.
Click to pick the measure starting point and then click to the end point. Move the cursor to seek the desired position of the dimension and then click when you found it.
Hold the Alt key to axis-align the measure.

Angle [*Alt+5*]
To add a measure angle First click to pick the vertex of the angle measure, then click to pick the first side (a half-line will appear) and finally click to pick the second side. Move the cursor to seek the desired position of the dimension and then click when you found it.

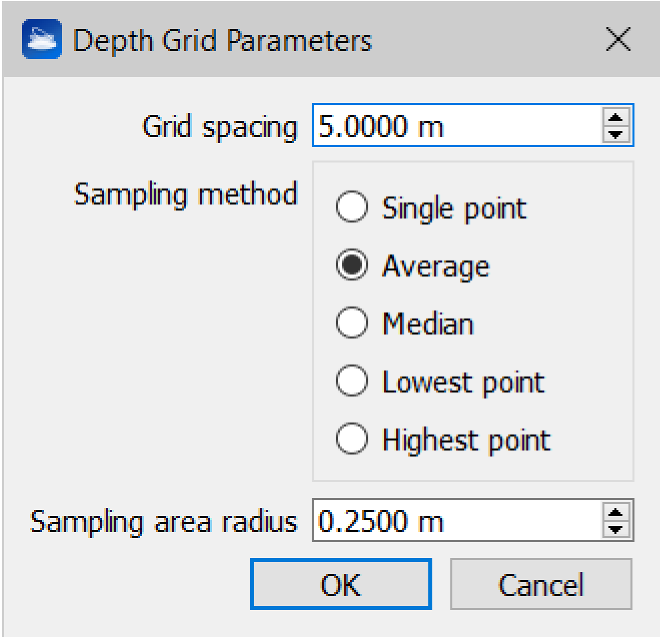


Area [*Alt+6*]
To define a perimeter on the map and compute the area inside it.
Click to pick the first vertex.
Click to add polygon vertices until you complete your perimeter and then click the first vertex again to close the area (that will be colored in red).
Move the cursor to seek the desired position of area's value annotation and then click when you found it.
During the picking point press Esc to delete the latest point.

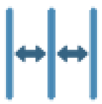
Volume [*Alt+7*]
Drawing a planar area on the orthophoto with a guided polyline selection, the volume identified by projecting this area along a user-defined depth is computed.

Depth
If the Depth computation is performed during orthophoto creation, a labeled point is added to the orthophoto's measures by clicking on a chosen pixel in the image. The label indicates the value of the depth (or altitude) referred to the pixel position.

Depth Grid
If the Depth computation is performed during orthophoto creation, a labeled grid is added to the orthophoto's measures: the *Grid spacing*, the *Sampling method* ("Which point you want to extract from the 2D image?" "The lowest in altitude to create a DTM", for example) and the *Sampling area radius* (the average value is computed within the circle defined by a range of influence around the node) are set in the *Depth Grid Parameters dialog*



while the position of the grid is defined by clicking on a chosen pixel in the image.
The labels on the nodes of the grid indicate the values of the depth (or altitude) according to their position.

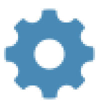


Continue
If pressed after the *Linear/Aligned/Angle* commands, permits to add other dimensions, aligned to the latest one.

After inserting the dimensions, they will be also listed in the Project Window as items related to the orthophoto.

Delete
To delete the selection of objects made by the *Selection* tool.

Settings



Settings
To set the viewer settings and the properties of the 2D image.

Viewer Settings
Check/uncheck *Show tools instructions* to show/hide tips on how to use commands use and on orthophoto properties.
Check/uncheck *Draw grid* to show/hide the grid. Here you can also choose the grid colorization.
Note that the grid unit is depending on the zoom level.
By clicking on the Change units... button it's possible to change the current units of measure. Few preset are available, but different units of measure for lengths, areas and angles can be set, as well as the number of decimal digits.

Orthophoto (X-Ray) properties
Here the background (and foreground if you are managing a X-ray orthophoto) colors can be changed.

Also the color (RGB and alpha channel), the arrow size and the text size of the dimensions can be changed as you please.
The image resolution is displayed, as well as the total photo (image) size in pixels.
If the resolution is here changed, it's necessary to recreate (snap again) the orthophoto in order to save the changes.

Recreate
To regenerate the orthophoto with the currently visible point clouds. In case of X-Ray orthophotos, the system will ask you which clouds you want to include in the regeneration.

Origin
To set a new origin of the orthophoto by double clicking on an orthophoto's pixel to set the axes and depth origin. Double right click to reset axes and depth origin.

Export

Export
To export the orthophoto (including all the customized dimensions) in different formats:

1. Image file
A *.png file is exported, keeping the same dimension and resolution of the original orthophoto.

2. AutoCAD Script
Several options are available to export your orthophoto to AutoCAD, in the settings dialog.
See Exporting orthophotos to AutoCAD for more details.

Note

If changes are applied from the last time you open the orthophoto viewer, you are asked to redo the snap.
The Logarithmic Transform *is suggested as contrast enhancement algorithm: the greater is the point cloud density in the view direction, the greater is the contrast (according to a logarithmic function).*

Create Camera

This sub-menu contains functions to create

Perspective

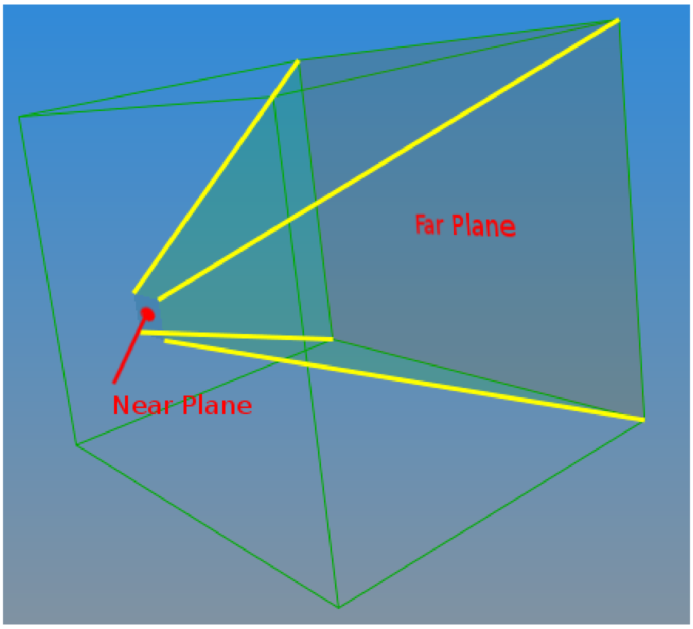
Orthographic

Cylindrical

Spherical

cameras with origin in the current point of view (the center of the 3D rendering window in your monitor).

Note: Perspective Camera



A perspective camera can be used to model a common photographic camera (e.g. a reflex camera), which follows the rules of perspective projection.
This type of virtual camera can be used to acquire a virtual scan or to define a point of view in the scene, which can be fixed by right clicking on the camera item inside the project tree and selecting the apply projection command.

This camera can be converted into a perspective projector by setting an image using the set projector image command. Note that the image to be applied needs to be undistorted.

This camera can be also used ti fit Orthophotos or X-Ray Orthophotos.

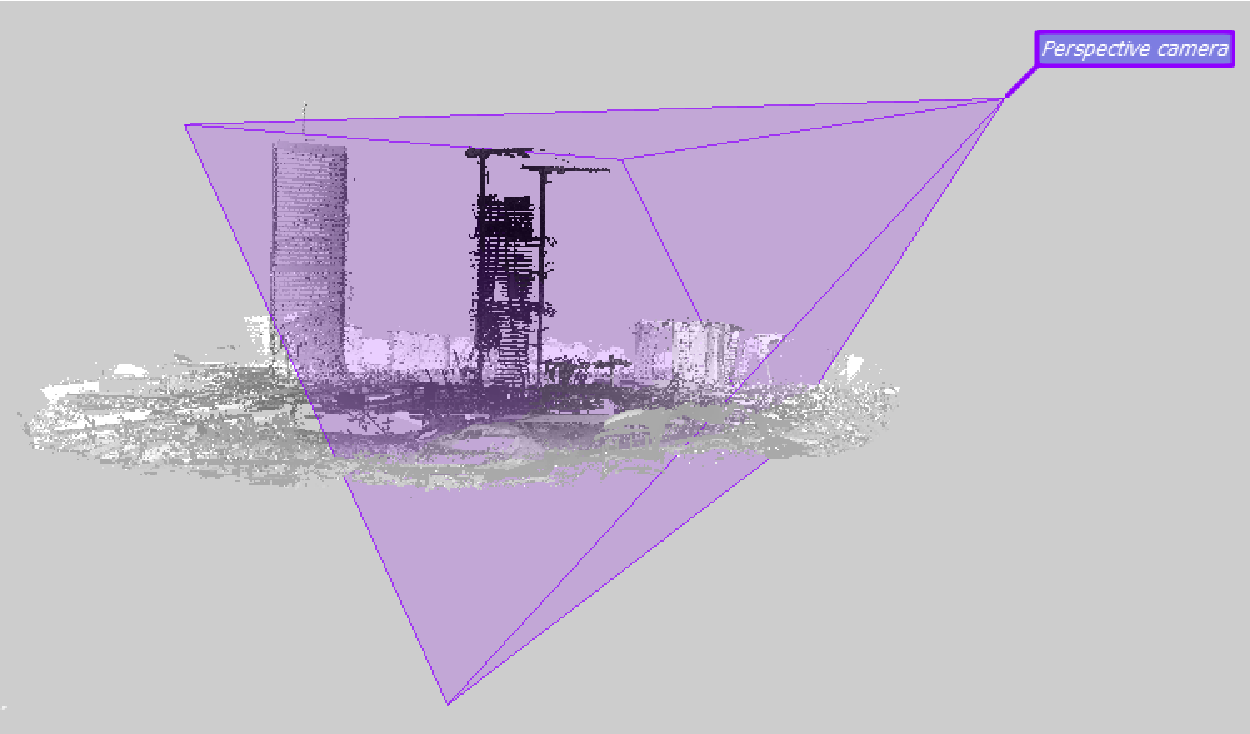
See Cameras for more informations regarding perspective cameras properties and associated functions.

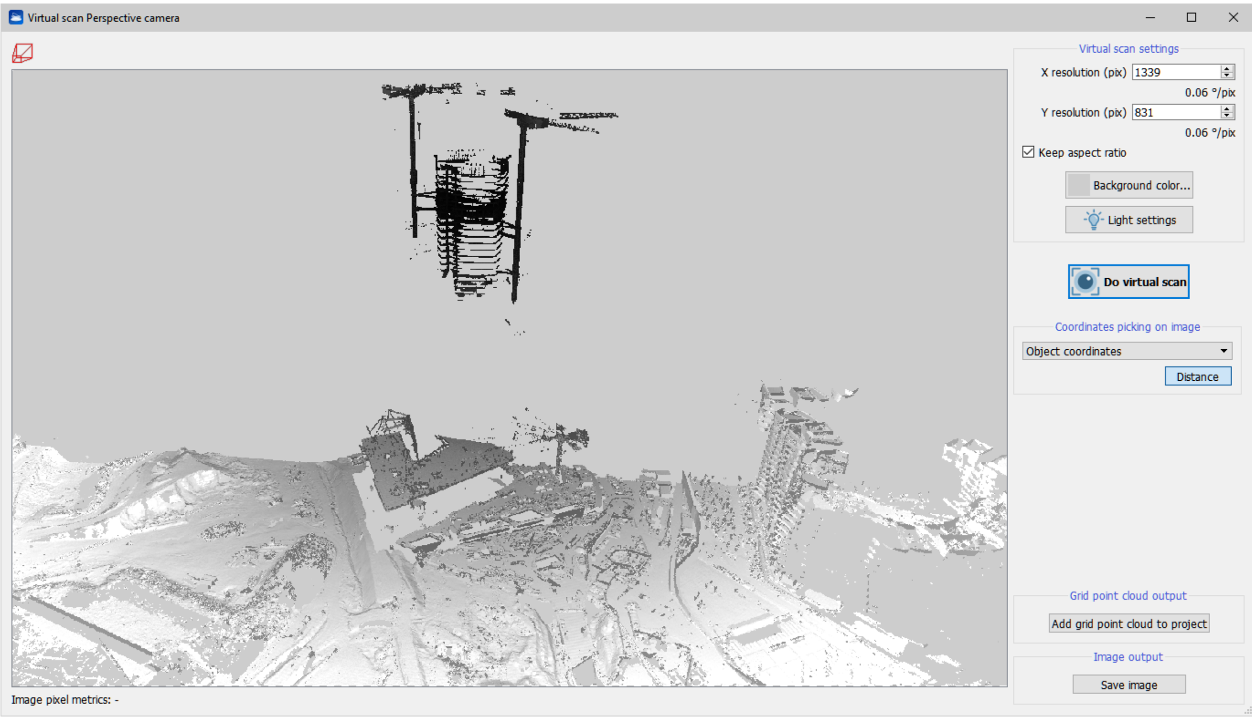
Place Here Survey Point

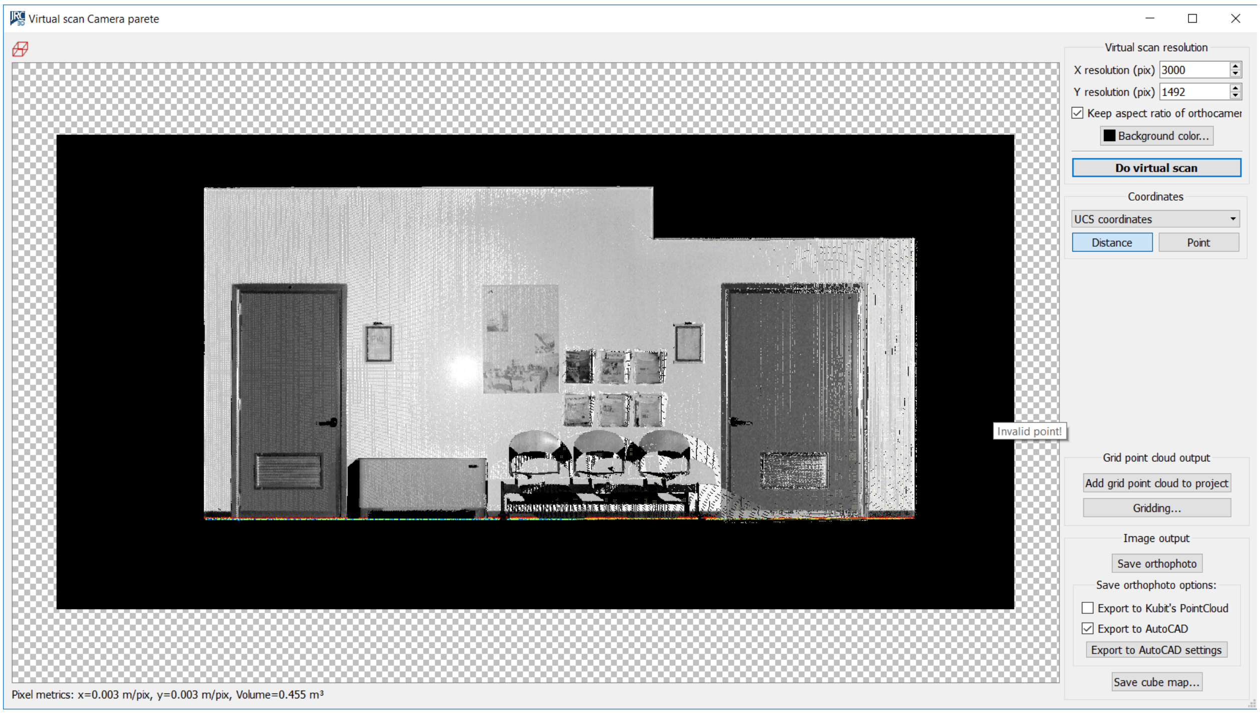
perspective
This tool creates a camera with origin in the current point view and current View Parameters settings.

Virtual Scan

This function is called *Virtual Scan* because you can imagine it like a scanning of the loaded models in the 3D window with a virtual laser scanner (a virtual camera).
In addition you can set up the near view plane (from which to start viewing the objects) and the far one.


Perspective camera


Virtual scan Perspective camera
Virtual scan settings
X resolution (pix) 1339
0.06 °/pix
Y resolution (pix) 831
0.06 °/pix
Keep aspect ratio
Background color...
Light settings
Do virtual scan
Coordinates picking on image
Object coordinates
Distance
Grid point cloud output
Add grid point cloud to project
Image output
Save image
Image pixel metrics: -



This function is used to resample the 3D dataset on a 2D grid from a defined position in 3D space and with a defined projection (orthogonal, perspective, cylindrical, spherical). Checks the models that must be scanned in the Project Items Window, and the depth range of the scan (near and far clip planes) of the current camera. Can create snapshots, cube maps or grid point clouds.

Parameters:

- **Update preview**: creates a preview of the scan from the selected view point and projection. Select the resolution and background color first.

Pick mode:

- **Distance**: press Left Mouse Button (LMB) for the first point and keep it pressed while dragging the mouse to the second point, then release the LMB. Both endpoints must be valid 3D points!
- **Point**: double click LMB to pick a list of 3D points from the virtual scan. If the AutoCAD link is enabled, the points are sent to AutoCAD. Check Global Coordinates for reading global/local coordinates.

Save cube map: (orthographic camera only) use this view point to construct a cube map of the scene, i.e. 6 square images (the faces of the cube) of size Width with perspective projection (90 degrees vertical field of view) are generated around the centre of projection and saved to file.
Save image: save the current virtual scanned image to file.
Save grid point cloud: create a grid point cloud from the current virtual scanned color and depth. The model is automatically added to the project.
The bottom row of the dialog shows the unit scale (depends on the type of projection) of the image and the computed volume between the plane of projection and the visible 3D data. For best results use triangle meshes, so no holes are found on the surfaces. Otherwise, try to increase the point size of the cloud of points in the Property Editor.

Orthophotos

A virtual scan of a plane/ortho camera or projector will be an orthogonal projection and thus an orthophoto. If the image is saved with Save image, a text file is created along with the image file

which exports registration information of the ortho image in the scene.
Simultaneously with saving the image a script for AutoCAD® is generated. It allows, by simply drag&dropping in the AutoCAD® window, to load the orthophotos in the correct position and scale, (in a blue layer with the same name dedicated to image). It also creates a UCS having XY drawing plane coincident with the image plane and positioned in the lower left side of the orthophoto, in order to facilitate the user in redrawing of vectorial draw.
Press the Export to AutoCAD settings button to choose the options.
Remember that the DWG file (template acadiso) must be saved in the same folder and the script files (* .scr).

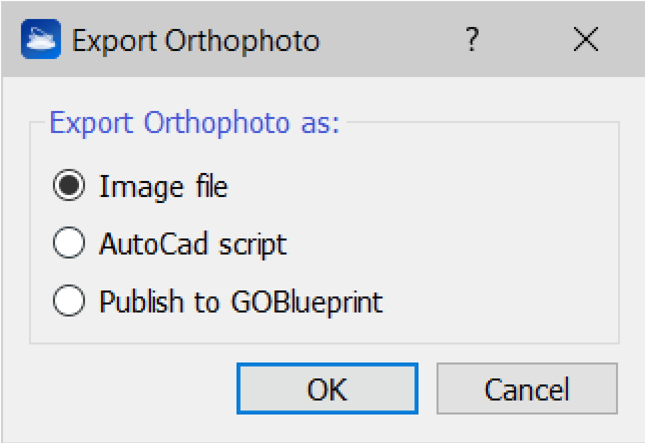
Gridding: this process computes the optimal estimation for missing points in a grid of size MxN, given K valid initial points. The algorithms available are: simple kriging (user has to specify the global mean), ordinary kriging (the mean is computed automatically), kriging with trend (the mean varies smoothly). Gridding is a demanding process, the time complexity is $O(K^2MN)$, so use as few as possible initial points.

Export 3D position and orientation of orthophoto to AutoCAD: to choose whether you want to export to AutoCAD the plain 2D image, or whether also the 3D position and orientation that the image has in Reconstructor®'s current UCS should be exported to AutoCAD.
Create new layer in AutoCAD containing the orthophoto: to choose whether to export the orthophoto in a new AutoCAD layer or in the current layer. Depending on the number of layers in the AutoCAD project, you may want to add a new layer or not.
Create new UCS in AutoCAD with same position and orientation of orthophoto: to choose whether you also want to create in AutoCAD a new UCS with the same position and orientation of the image, or not. If you selected to export the 3D position and orientation as well, then this checkbox is enabled.
Suggested steps for orthophoto export to AutoCAD

- Click on *Save Orthophoto* button: an image (.png suggested) and a script file (.scr) will be created.
- Create and Save a .dwg file in the same folder of the exported orthophoto (acadISO format suggested)
- Drag&Drop the script file in AutoCAD dwg file just created: the orthophoto will be automatically positioned in the 3D space if you flagged *Export 3D position...* in *Export to AutoCAD settings*, otherwide you need to define the origin of the orthophoto in the XY plane of the current UCS.
- If you need to reset the orthophoto, you have to "Detach" it, save the AutoCAD project and drag&drop the script file another time.

See also Create Orthophoto.

Exporting orthophotos to AutoCAD

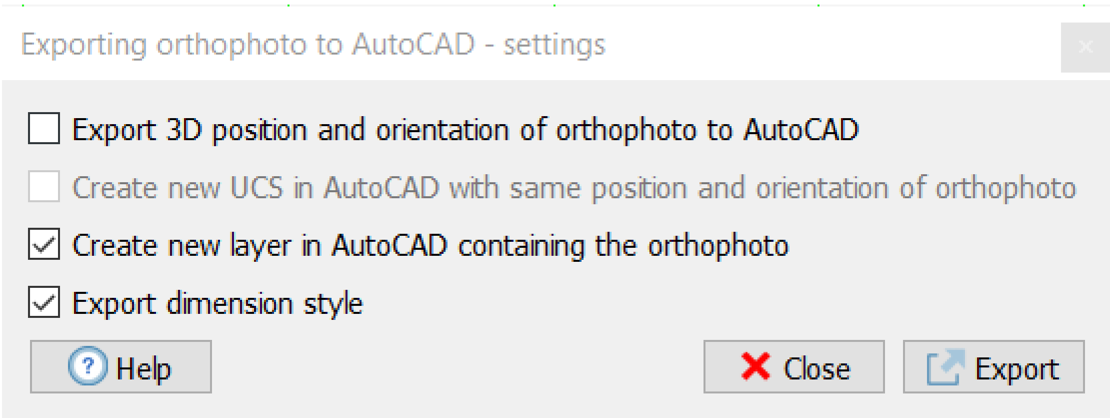


1. Image file

A *.png file is exported, keeping the same dimension and resolution of the original orthophoto.

2. AutoCAD Script

Several options are available to export your orthophoto to AutoCAD, in the settings dialog.



Export 3D position and orientation of orthophoto to AutoCAD: to choose whether you want to export to AutoCAD the plain 2D image, or whether also the 3D position and orientation that the image has in Reconstructor®'s current UCS should be exported to AutoCAD. Exporting the image's 3D pose to AutoCAD is useful if other items (e.g. polylines) related to the same model are exported to AutoCAD.
If you selected to export the 3D position and orientation as well, then the second checkbox is enabled.
Create new UCS in AutoCAD with same position and orientation of orthophoto: to choose whether you also want to create in AutoCAD a new UCS with the same position and orientation of the image, or not. If you selected to export the 3D position and orientation as well, then this checkbox is enabled.
Create new layer in AutoCAD containing the orthophoto: to choose whether to export the orthophoto in a new AutoCAD layer or in the current layer. Depending on the number of layers in the AutoCAD project, you may want to add a new layer or not.
Export dimension style: to give choice about whether to export also the dimension style (AutoCAD compatible) or not.

Check this option only for the first import in AutoCAD.

Please, note that the dimensions are saved in a separated dimension style in AutoCAD called *Reconstructor*.

If two first options are flagged, at the end of the import process in AutoCAD the view is focused on the origin of local orthophoto's UCS, to fastly localize the orthophoto.
To visualize a georeferenced orthophoto in the correct global reference system (according to global coordinates), set the World UCS *in AutoCAD.*

Click on *Export* button to automatically save an image (*.png format) within together a script file (.scr).

Suggested steps for orthophoto export to AutoCAD

1. Click on Save Orthophoto button: an image (.png) and a script file (.scr) will be created.
2. Create and Save a .dwg file in the same folder of the exported orthophoto (acadISO format suggested)
3. Drag&Drop the script file in AutoCAD .dwg file just created: the orthophoto will be automatically positioned in the 3D space if you flagged *Export 3D position and orientation of orthophoto to AutoCAD* in Export to AutoCAD settings, otherwise you need to define the origin of the orthophoto in the XY plane of the current UCS.

If you need to reset the orthophoto, you have to "Detach" it, save the AutoCAD project and drag&drop the script file another time.

To set a transparent background set the alpha channel to 0 (zero) in Orthophoto Viewer Background color options.

Cross Sections

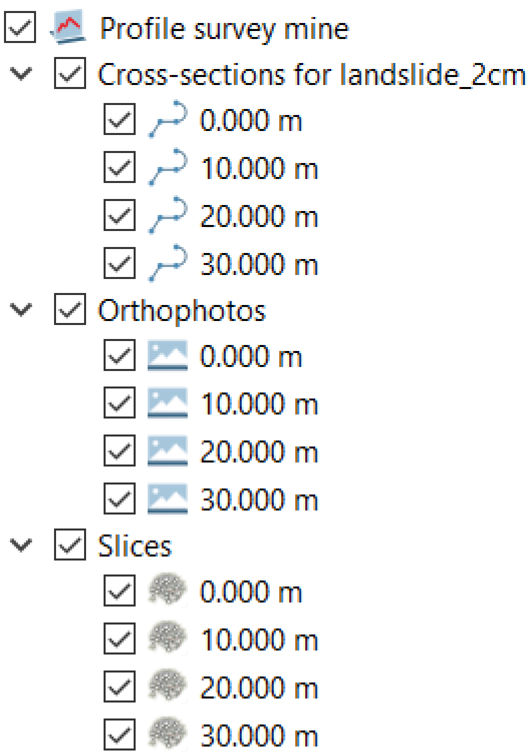
This submenu contains commands and features to create cross sections and isolines of point clouds (grid or unstructured) and meshes, starting from cutting planes and other constraints:

- Create/edit plane
- Cross sections
- Contours
- Quick profile
- Cross sections from planes
- Section from plane
- Tunnel cross sections

Note that only 3D models (point clouds or meshes) loaded and displayed in the 3D window will be considered in the sectioning processes.

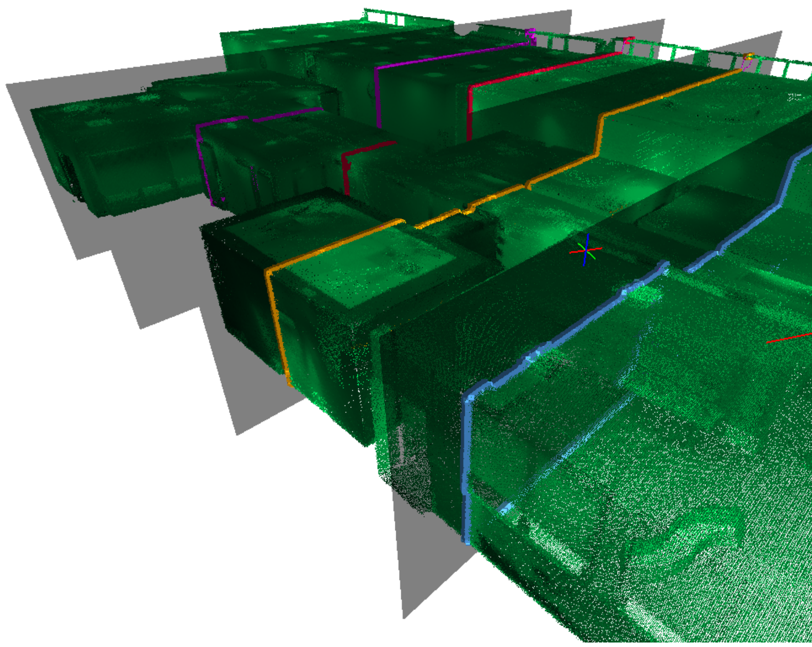
Advanced cross sections structure overview

Section from plane, Contours, Quick profile and *Tunnel cross sections* tools give as output a special **Survey** item composed by one orthophoto, one cross section polyline and one slice of point cloud (*if required and if sectioning point clouds*) for each extracted section.



These items are merged in groups named

- *Cross Section for ...*
- *Orthophotos*
- *Slices*



Note that when sectioning more than one 3D model, one group pe

will be created including the polylines sections concerning that mo
group will be indeed created both for Orthophotos and Slices items.
to create any single Slices item a merging of different point clou
done.

Several features are common in *Section from plane, Contours, Quick profile* and *Tunnel cross sections* tools. They are explained below.

Navigation across sections

During cross section display a preview of the final cross sections, any navigation modalities are available in the recipe window.

Map view

This navigation mode places the view so that all the selected items can be seen from a top view.

Default view

This navigation mode allows the user change the current view so that all the selected items are completely visible in the 3D scene.

Section view

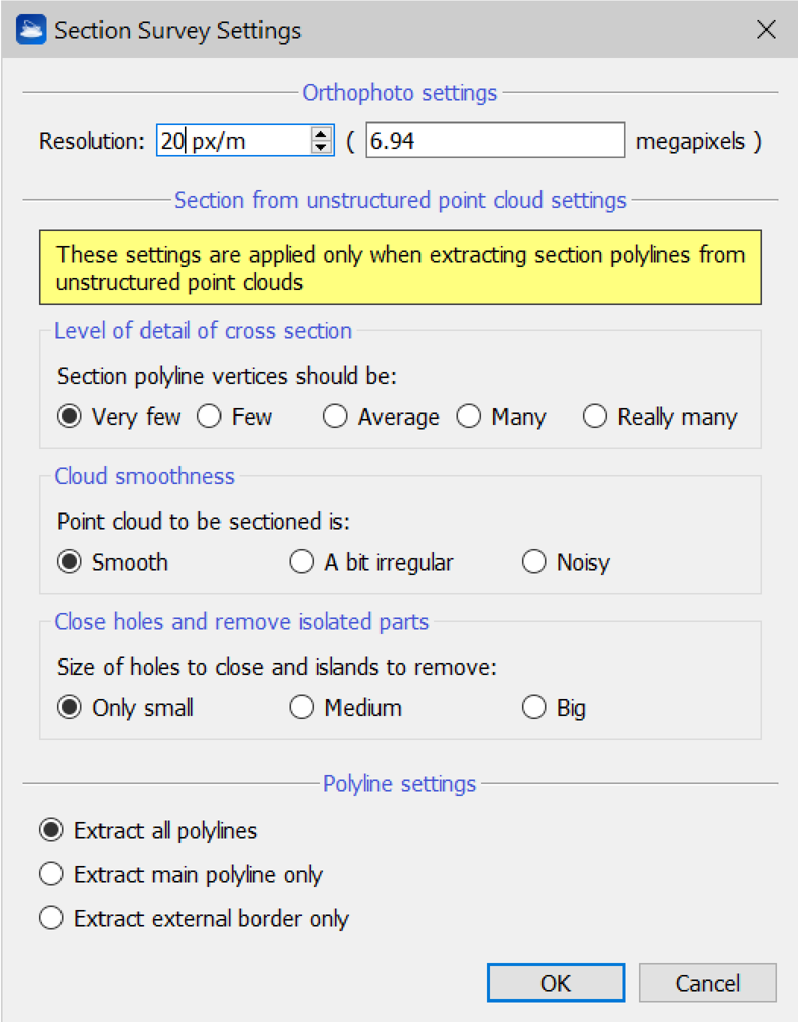
This navigation mode automatically switches the view to orthographic mode and aligns it to the cross section view.

Left and right arrows

The arrows can used to switch from a cross section preview to another.

Section Survey Settings

By clicking button the settings parameters can be changed to manage the orthophotos' resolution and quality of extracted polylines.



Orthophoto settings

The resolution is automatically computed according to orthophoto's dimensions, but it is possible to manually change it. The resulting dimension (in megapixels) will be computed and displayed.

Section from unstructured point cloud settings

When sectioning unstructured point clouds further settings can be used to extract more detailed cross sections polylines.

Level of detail of cross section: you can specify how many vertices you want to use for your output polyline. The more the vertices, the finer the details that will be reconstructed.

Cloud smoothness: if your cloud represents clean construction data, then select *Smooth*. If, however, your cloud contains noisy data, then select *Noisy*.

Close holes and remove isolated parts: you can here specify the size of the holes that are going to be closed in the polyline, and the size of the “islands”, small disconnected components that may have been created. The size is computed as ratio between the length of the hole/island and the length of the whole polyline.

Polyline settings

The cross sections polylines are computed interpolating the slice of point cloud at the turn of the cutting planes.

By selecting one of the following options you can define the degree of extraction of cross sections polylines to be included into the final section survey.

These settings must be mainly managed in tunnels sectioning.

Extract all polylines: to keep all the edges obtained from the

intersection between cutting plane and model's slices.
Extract main polylines only: to delete short edges from polyline.
Extract external border only: to close holes in main borders, keeping only most external borders. Be careful when more tunnels or hollows are intersecting in the same section.

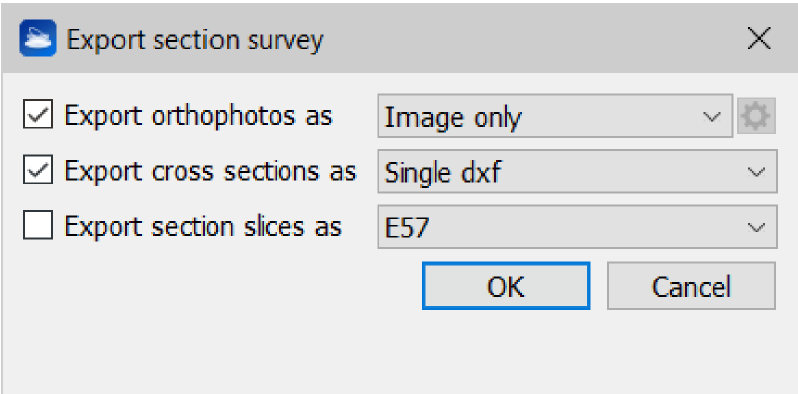
Compute
After managing cutting planes and parameters click on *Compute* to extract the sectioning outputs.

After creating a Section Survey, it will appear in the Project Window items list.

You can access the results at any time by the Survey's context menu, clicking on *View cross sections* command: the Orthophoto viewer also displaying polylines and slices will be opened.

Export Section Survey

All the created items can be exported individually through the traditional export commands.
In the Survey's context menu another possibility is available: through the *Export section survey* command you can export all the objects in a single step.
To do it, flag and define the exportation options, according to your purposes.



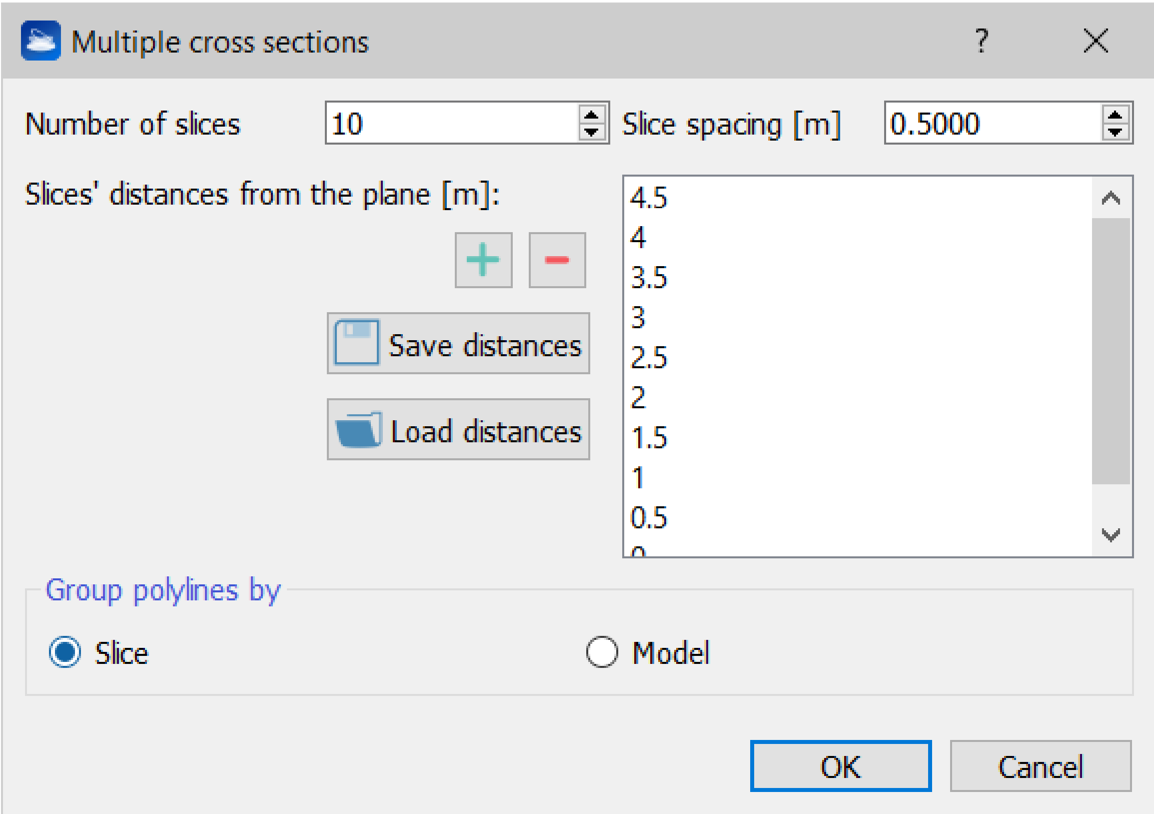
Export orthophotos as
Image only: to export only an image file, not scaled and not georeferenced
Image + AutoCAD script: to export an image file within a script file to import scaled and georeferenced images in AutoCAD (see also Exporting orthophotos to AutoCAD)

Export cross sections as
Single dxf: to export cross sections as polylines where any group of polylines will be included in a single .dxf file
Multiple dxf: to export all the cross sections as polylines in a single .dxf file
Multiple Shape files: to export all the cross sections in a single .shp file

Export section slices as *E57/LAS*: to export the slices as point clouds in E57 or LAS format.

Cross section

To create a ***cross section*** of your models you can access the relative command in the top toolbar or from the planes' context menu.



You can make cross sections of grid point clouds and meshes. However, it is not possible to make cross sections of unstructured point clouds.

Cross sections are defined with respect to a plane.
With the default settings, the section is defined as a polyline created by cutting all loaded models with the plane. However, it is also possible to have *multiple cross sections* by defining more slices or additional cutting planes, all parallel to the original plane and lying at a certain distance from it.

In the top left spinbox you can specify how many *slices* you want, the default is one. The top right spinbox allows you to specify the default inter-slice distance. By editing these two parameters, you will notice that the values in the panel on the right in the dialog are changing. These are the list of the slices' distances from the plane. By double-clicking on a single distance, it is possible to edit individually each distance. Furthermore, it is possible to add (green “plus” button) and remove (red “minus” button) the currently selected slice. There are also two buttons to *Save the distances* and *Load the distances*: these are useful when you have to use the same distances set for many cross sections.

Distances are defined on the plane's *Z* axis, therefore 0 means “slice lying on the plane”, and the positive direction is given by the direction of the plane's *Z* axis. You can invert the plane's direction with the appropriate command in the plane's context menu. It is also possible to have negative distances, by double-clicking on them and writing a negative number.

Towards the bottom of the dialog there is the *Group polylines by* panel, with the option to either group the output cross sections by *Slice* or by *Model*. With the first option, one cross section will be created for each slice, regardless of how many models the slice crossed. With the second option, only one cross section will be created for each model crossed, regardless of how many slices crossed that model and where.

It is possible to use this last modality to create isolines (contour lines).

When you press *Ok*, the specified cross sections are computed and added to the project under the *Cross sections* items group. Cross sections can be exported in DXF for AutoCAD.

Contours

To create ***isolines (contour lines)*** of your models (grid or unstructured point clouds and meshes) you can access the relative command in the *Mining* top toolbar.

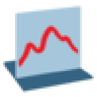
According to the frustum of the loaded models in the project, several horizontal cutting planes (parallel to X and Y axes of current UCS) will be created; it's possible to define starting and ending altitudes and the interval between planes. These planes will be automatically used to intersect the models and extract the contour lines.

Flag *Extract point slice* to extract also slices of point cloud in correspondences of the each isoline. The thickness of the slice is defined by the slice *depth* value.

It is really suggested to use this tool on continuous and without opened boundaries surfaces.

See more details about the sectioning tools and export modalities at Cross sections introduction.

Quick profile



To create ***vertical cross sections*** of your models (grid or unstructured point clouds and meshes) by simply picking two points in the 3D window you can access the relative command in the top toolbars.

After clicking the *Pick profile* button, double click on two points in the 3D scene define start and end points of the section line; select ✔ to accept the choice. A cutting plane is created based on this profile, but its width and height can be changed later.

Flag *Repeat profile* to create more profiles, setting intervals and number of sections parallel to the main one.

Flag *Extract point slice* to extract also slices of point cloud in correspondences of the each profile. The thickness of the slice is defined by the slice *depth* value.

See more details about the sectioning tools and export modalities at Cross sections introduction.

Cross sections from planes

This command creates a list of cross sections starting from a sequence of planes.

To extract multiple cross sections from different planes in a single step:
1. In the Project Windows, select the planes you want to use to cut the 3D models loaded in the project
2. Open the context menu by clicking the RMB
3. Select the *Cross Sections* > *Cross sections from planes* command

Section from plane

To create ***cross sections*** of your models (grid or unstructured point clouds and meshes) by selecting a cutting plane you can access the relative command in the *CAD Output* top toolbar, below the *Cross sections* tools.

After creating a plane you can select it through the *Select plane* button. You can here set its width and height dimensions, independently from the original ones.

Flag *Repeat profile* to create more profiles, setting intervals, number of sections parallel to the main one and direction of generation of cutting planes parallel to the main one (above, below or on both sides of this plane).

Flag *Extract point slice* to extract also slices of point cloud in correspondences of the each cross section. The thickness of the slice is defined by the slice *depth* value.

It is really suggested to use this tool on continuous and without opened boundaries surfaces.

See more details about the sectioning tools and export modalities at Cross sections introduction.

Tunnel cross sections

To create ***cross sections*** of your models (grid or unstructured point clouds and meshes) following a trajectory you can access the relative command in the *Mining* top toolbar, below *the Tunnel Survey* tools.

The path used to extract multiple cross sections can be defined by points directly picked in the 3D scene, or by polyline and trajectory already available between the project's items.

After clicking the *Points* button, double click on several points in the 3D scene to define the trajectory of the the axial section line.
Select the *undo* button to delete the last picked point.

After clicking *Polyline* or *Trajectory* a list of available project items appears to invite to select the desired one.

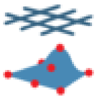
Select  to accept the choice or  to deny it.

Several cutting plane are created based on this profile, but its width and height can be changed later.Flag *Repeat profile* to create more profiles, setting intervals and number of sections parallel to the main one.

Flag *Extract point slice* to extract also slices of point cloud in correspondences of the each section. The thickness of the slice is defined by the slice *depth* value.

See more details about the sectioning tools and export modalities at Cross sections introduction.

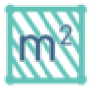
DEM points



This function allows you to extract simplified models of your models (grid or unstructured point clouds and meshes) you can access the relative command in the *Mining* top toolbar.

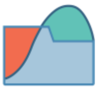
Areas & Volumes

These tools compute area and volume of point clouds and meshes portions.



Area
To compute the area of portions of point cloud and meshes surfaces.

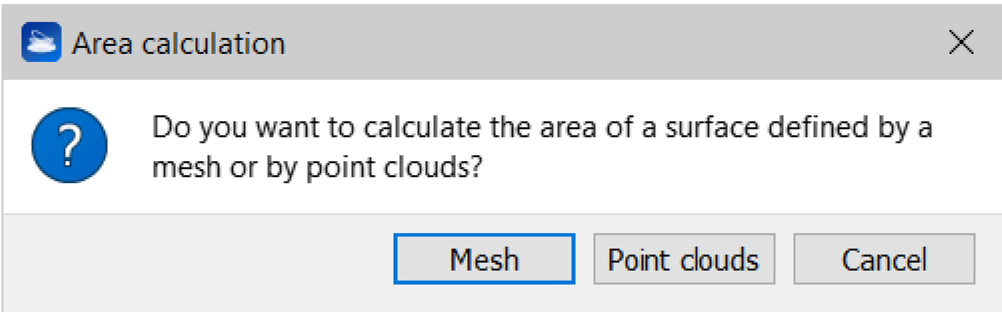
Volume from plane
To compute volume of space included between a mesh surface and a plane.



Cut&fill volumes
To compute cut and fills volumes by comparing two meshes.

Area

Reconstructor® provides different ways of computing areas.

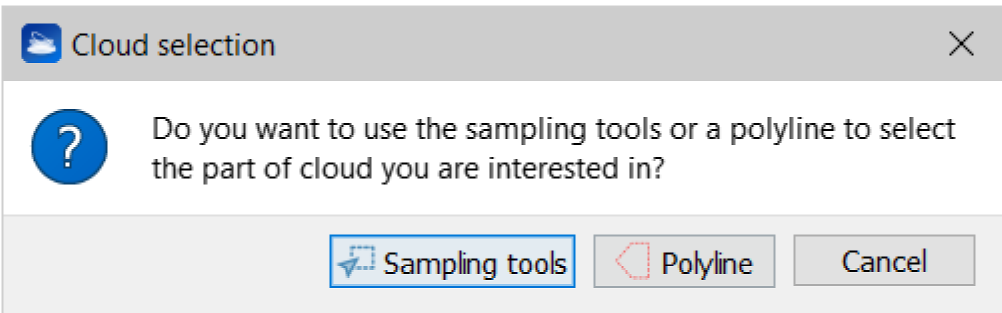


When you press the *Area* button you can choose if you want to compute an area defined by a mesh (1) or by a point cloud (2).

1. Area of a mesh portion

By using the Sampling tool you can perform a selection of a mesh's portion A new mesh is created using only the geometry visible from the current view point.

2. Area of a point cloud portion



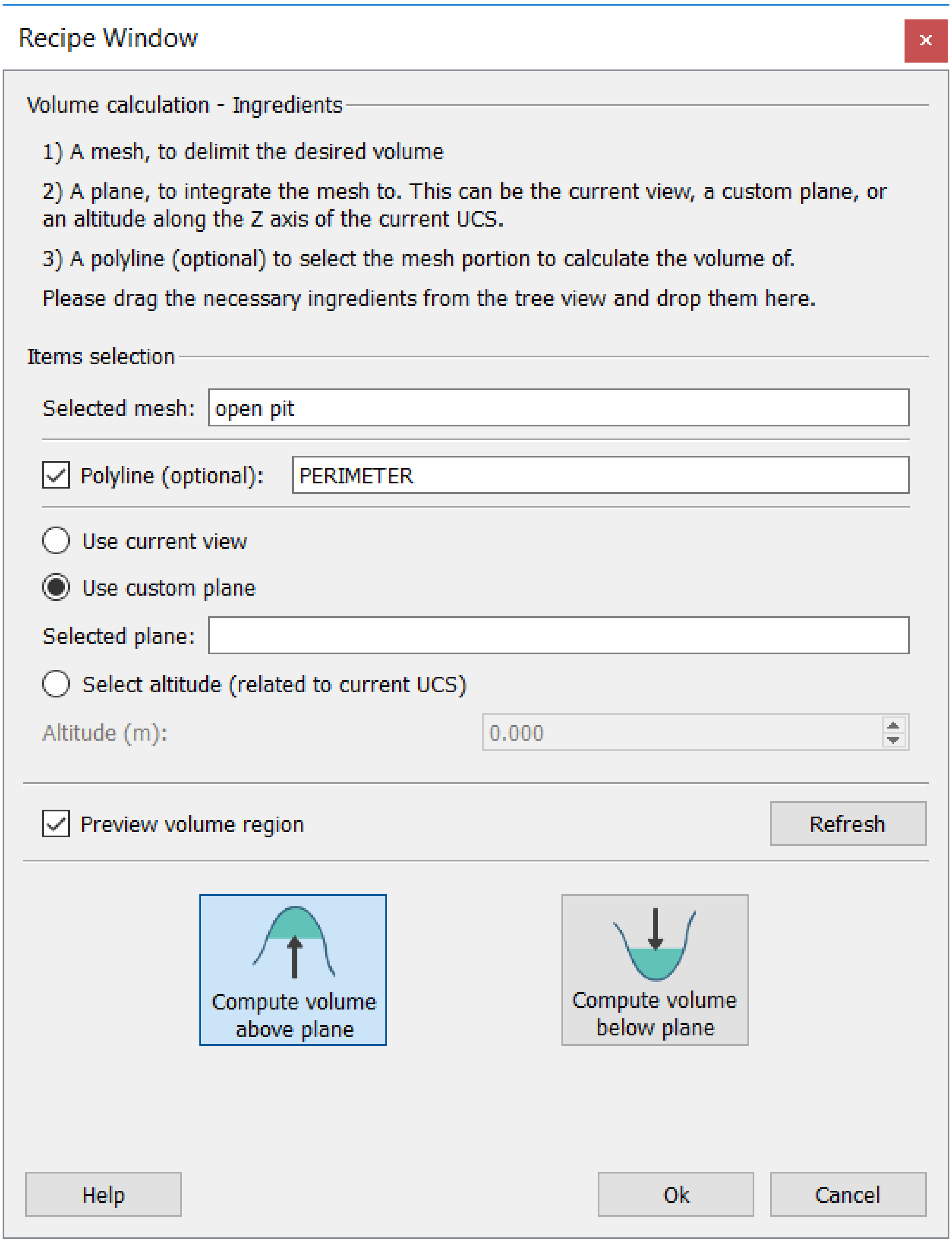
- *Selected with Sampling tools*

You can select the point cloud portion by using *Sampling tools* on the 3D window. This selection can span across multiple point clouds.

- *Selected with 3D polyline*

You can select the point cloud portion by using the Point selection with polyline command.

Volume from plane



This procedure enables you to calculate volumes by integrating meshes, or portions of meshes on a custom plane. You can activate this tool via *Outputs->Area & Volume Volume* in the main menu, or via the same button in the top toolbar. On activation, the window above appears in the *Recipe window*.

You define the mesh, the plane and optionally the polyline to use with the usual drag-and-drop mechanism. Reconstructor® will calculate the volume of the mesh integrated onto the plane, regardless of the triangles' normal directions. It should be noted that only the portion of mesh lying in the *positive* semispace of the plane will contribute to the volume. Parts of mesh lying below the plane will result in zero volume.

Optionally, you can also provide a polyline. The polyline, projected on the plane, defines a closed polygon on it. If you provide a polyline, only the mesh portion whose projection falls inside that polygon will count for the volume. It is like if you would cut the mesh with the polyline before computing its volume.

You have three ways to define the reference plane: either using the current view, or dragging a custom plane, or using the horizontal plane with a given *Altitude* on the current UCS.

You can *Compute the volume above the plane* (and under the mesh) or *Compute volume below the plane* (and above the mesh).

If you check *Preview volume region*, then Reconstructor® will show a preview of the 3D region of which the volume will be computed. This preview refreshes if you change any of the parameters from inside the recipe window. It may not refresh if you change something from outside the recipe window, for example if you move the plane or the mesh with e.g. *Adjust pose*.

Once you have set all the parameters, press *Ok*. Reconstructor® displays a progress bar showing the calculation's progress, and at the end visualizes a short report on the volume measure. You have the options to copy this short report on the clipboard, or to access another window that enables you to produce an elaborated PDF report, via the option *Get PDF report*.

Volume PDF Report

This dialog enables you to create a detailed PDF report about your volume calculation.

The dialog allows you to fill in a series of fields with the information you want to appear in the report. The fields you uncheck or you left blank will not be included in the report.

On top left of the dialog you can fill the fields *Site name* and *Company name*. If filled, they will appear as title/subtitle of your report. Below, check *Survey information* if you want to include in the report information about survery operator and survey date. The checkable panel *Processing information*, below, works in the same way.

On the top right of the dialog, you can also select whether you want the current 3D view to appear as screenshot in your report. Finally, on the bottom right you can type additional notes to be included in the report.

The data you insert are saved also in Windows' registry, therefore the next time you use this dialog you will be able to reuse the data you filled in before.
When you have inserted the data of your report, click on *Save PDF report* and select a filename. The PDF file you specified will be generated.

If a cross section survey is found for current mesh model, an *Insert cross sections survey* tool is available.
If flagged, the cross sections related to a selected survey will be included in the PDF report, as orthophotos reproduction.

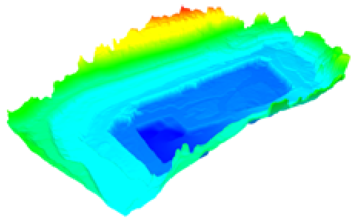
Open pit mine

Volume Report

September 23, 2020

Survey date: July 27, 2017.
Survey operator name: Gexcel.
Processing date: May 3, 2020.
Processing operator name: Gexcel.
Reference plane: "DTM base plane"
Surface integrated: DTM

The volume is 7558039.4441 m^3



Surface information

Triangles count: 1130481
Vertices count: 567033
Area : 281623.7247 m^2
Center coordinates: -223.6240 m 113.0174 m 100.7904 m

Cross sections information

Cross section survey name: Profile survey DTM
Distance between sections: 20.0000 m

Cross sections views

0.0000 m

Contents

- Survey and processing information, as Survey date, Survey operator name, Processing date and Processing operator name, ...
- *Reference plane*:plane used as reference surface to compute volume
- *Surface integrated*: mesh model used to compute volume above or below the plane

The volume is .... m^3 (computed volume between mesh model and reference surface)

A screenshot of the current view of the models is also included.

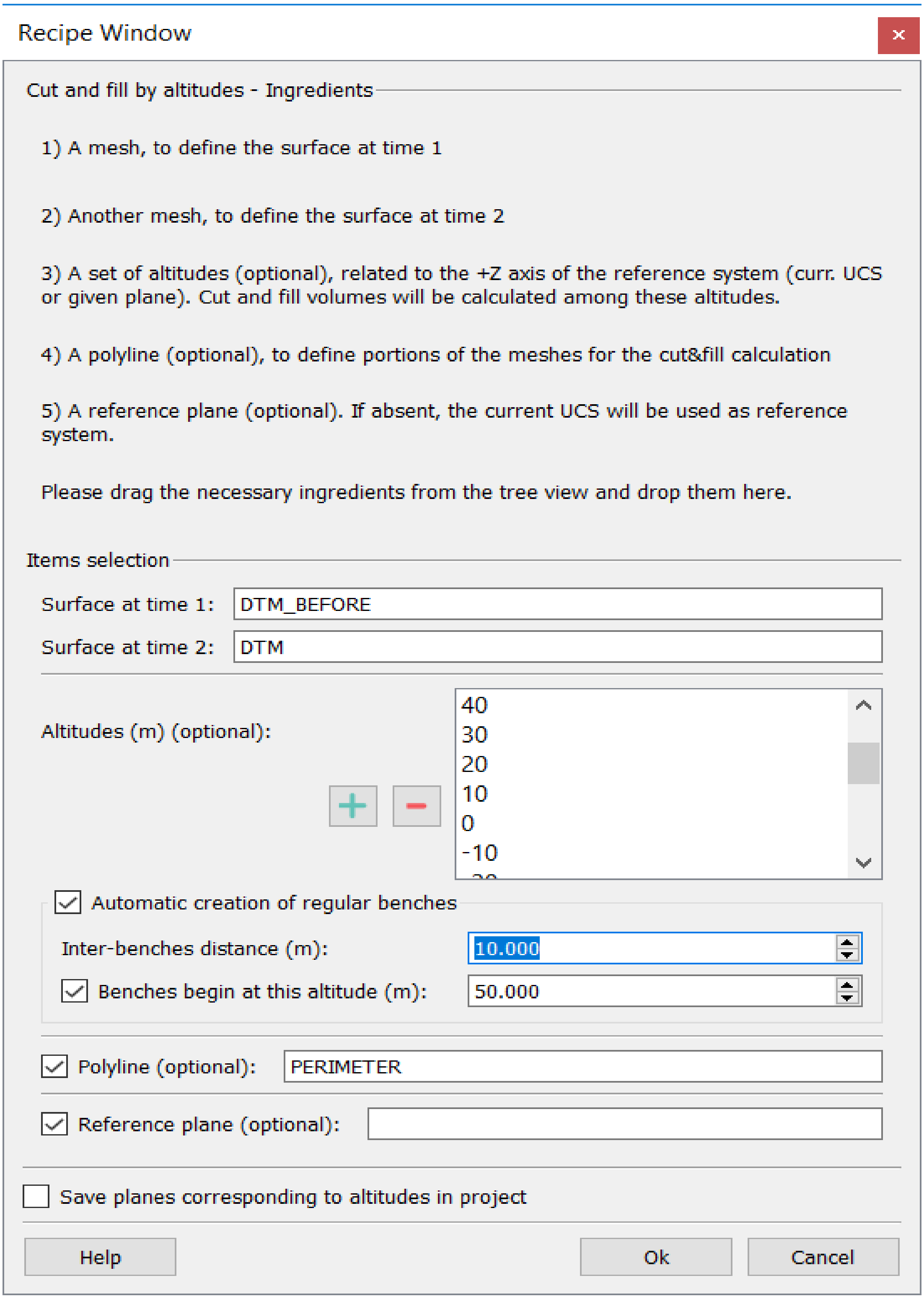
Surface information

- Triangles count: number of triangles of the mesh model
- Vertices count: : number of vertices of the mesh model
- Area : area of the mesh model's triangles
- Center coordinates: coordinates of the center position of the mesh model

Cross sections information

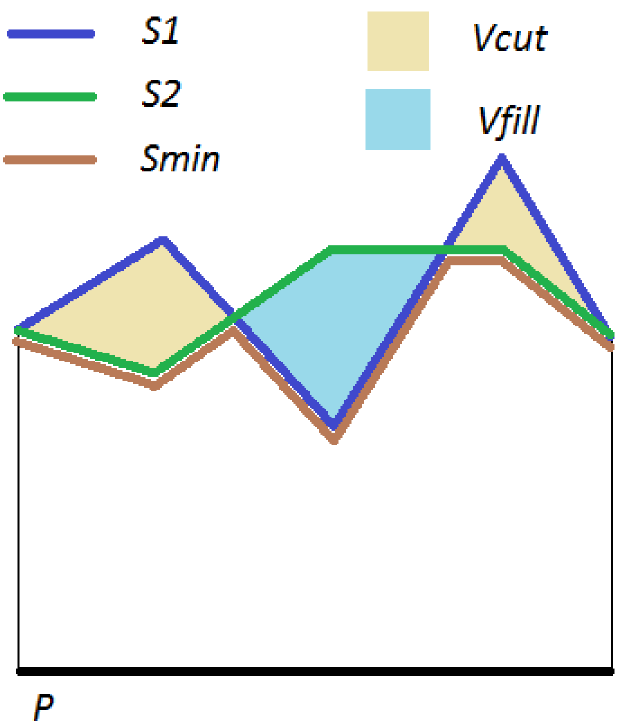
- Cross section survey name
- Distance between sections
- Cross sections views: list of 2D orthophotos displaying the extracted cross sections

Cut&fill volumes



This procedure implements calculation of *cut* and *fill* volumes. Given two meshes representing the same object (e.g. the same terrain) at different instants in time, the *cut* volume is the volume that the object *lost* between instant 1 and 2, the *fill* volume is the volume that the object *gained* between the two instants.

The simplest way to use this procedure is to drag in the recipe window one mesh representing the surface at time 1, another mesh representing the surface at time 2, and click OK. Reconstructor will return the volume of surface 1 integrated on the Z=0 plane of the current UCS, the volume of surface 2 integrated in the same way, the *cut* volume (by how much the surface was excavated between time 1 and time 2), and the *fill* volume (by how much the surface was filled (has gained volume) between time 1 and time 2).



Given two surfaces *S1* and *S2*, and a plane *P*, and assuming that the projections of *S1* and *S2* on *P* share a support *C* in common, a third surface *Smin* can be defined. Since each point in *C*, is the projection on *P* of (at least) one point of *S1* and one of *S2*, the one of these points that is closest to *P* is defined as belonging to *Smin*. Let *V1* be the integral of *S1* on *C*, *V2* the integral of *S2* on *C*, and *Vmin* the integral of *Smin* on *C*. Then the cut volume *Vcut* is given by *V1* - *Vmin*, and the fill volume *Vfill* is *V2* – *Vmin*.

The user can specify several optional parameters, to satisfy special requirements. A set of altitudes can be optionally defined, Reconstructor will calculate the contributions to cut and fill volumes for each bench among two neighboring altitudes. The altitudes can be added and edited manually. Add an altitude by clicking the *plus* button, the *minus* button removes the selected altitude. Double-click on an altitude to change its value. Otherwise, the altitudes can be generated automatically by specifying a starting altitude and a fixed bench height. Check *automatic creation of regular benches* to do that.

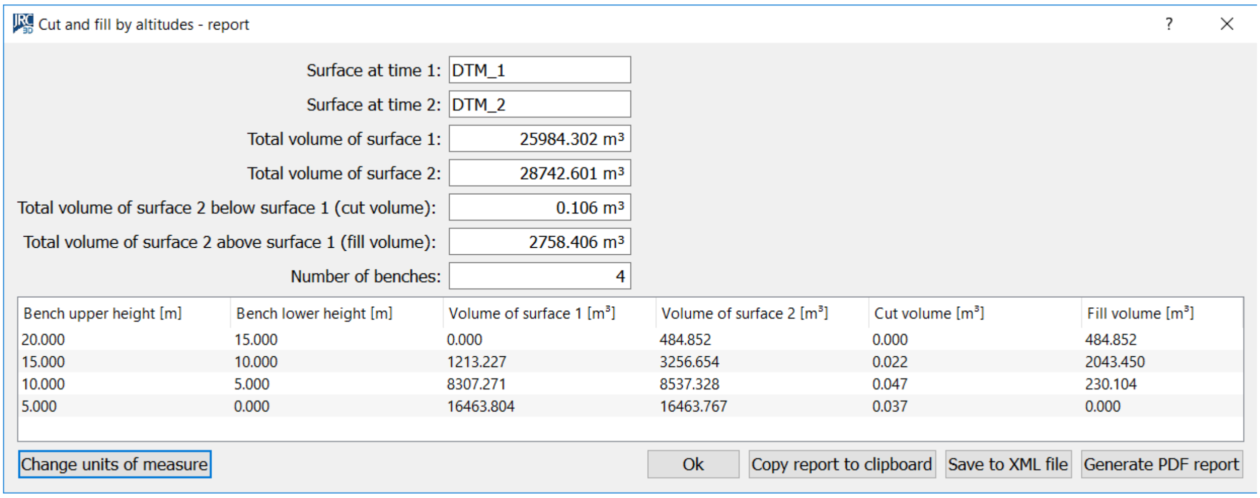
These altitudes are by default referred to the Z axis of the current UCS. However, the user can change this, by dragging and dropping a plane that fills the field *Reference plane (optional)* and becomes the reference of all cut and fill calculation and of all altitudes.

The last optional parameter is a polyline that the user can specify to delimit the support *C* on which the surfaces are integrated. The closed polyline is projected on the horizontal plane Z=0 of the reference system and the support *C* in common between *S1* and *S2* is intersected with the area delimited by the projected polyline. Therefore, the volumes are calculated only inside the frustum defined by the polyline and the horizontal plane.

Once you have set all the parameters, press *Ok* and the computation starts, monitored by a progress bar and by messages in the log window.

See also Cut&Fill report.

Cut and fill by altitude report



This dialog provides you a report of the results of a cut and fill calculation, along with options for managing and exporting the results.
From top to bottom, the names of the surfaces “before” and “after” are reported, then the volume of the surface “before” and then “after”, then the cut and fill volumes, then the number of benches considered for the calculation. Lastly, a table comes with the result displayed for each bench. In fact, you can define benches (intervals among altitudes) and see how the cut and fill volumes behave for each bench.
On bottom right, four buttons offer you options for exporting the results. You can copy the results in text form to the system clipboard, you can save the result to an XML file, or you can create a PDF report with the result. This report can be integrated with more data on company and site name, survey information and processing information. The button Generate PDF report opens theVolume PDF report dialog.
You can also *Change units of measure* displayed in the report.

Inspection

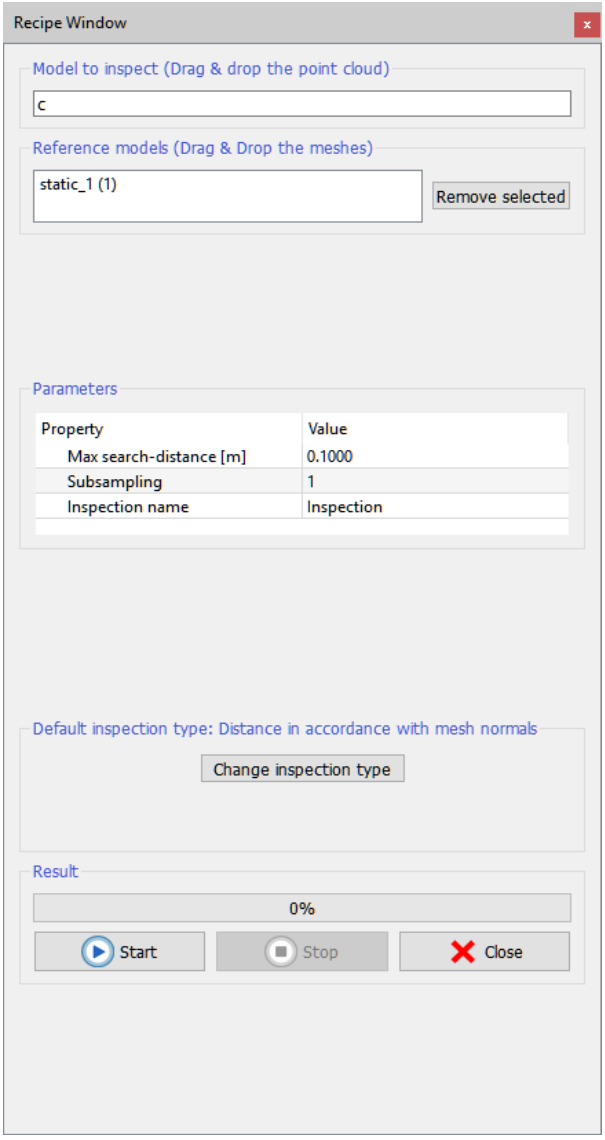
Inspection is a procedure to compare two different shapes and to measure their differences in geometry.
It is accessible from *Measures* tools in the top toolbar.

It is normally used to compare two models of the same objects surveyed at different moments in time. For example, it is used for monitoring barrels of nuclear waste to immediately detect possible deformations.

The result of the inspection is a new color layer that will be assigned to the point cloud to inspect. This color layer carries, for each point, its measured distance from the reference model. After performing *Inspection* tool it is possible to use the *Colors mapping dialog* function to map these measures with colors chosen by user and to see the corresponding color scale and histogram.

Two different processes are performed depending on the type of models to be compared: clouds and clouds/meshes.

1. Cloud to Mesh



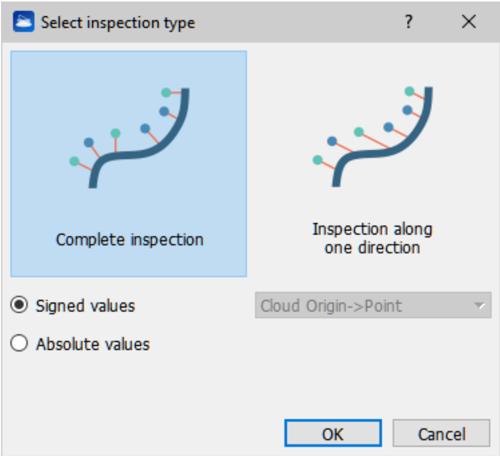
Input models

As inputs, user must provide a model *to inspect* and one or more *reference* models. The model to inspect must be a *point cloud*. The reference models must be *triangle meshes*.
The desired models can be inserted simply by dragging and dropping them. Press *Remove selected* to deselect undesired models from your import.

Parameters

- **Inspection name**: to define a name for the layer that will be that will be generated
- **Maximum search distance**: every point of the model to inspect will be compared with the reference model only inside a sphere with a radius defined by this parameter.
- **Cloud subsampling**: allows to balance speed versus precision.
  There are two cases:
  - *Inspect an unstructured point cloud*: the cloud will be divided in *s* sets of points and only one point in each set will be inspected. The other *s* – 1 points will be assigned the same distance value measured for that point.
  - *Inspect a grid point cloud*: the procedure will scan the points according to their structure. The grid's structure will be divided in cells of dimension *s* by *s*. Only one point per cell will be inspected and the other points will get the same inspection value measured for the first.

- **Inspection types**: to select the Inspection type to be performed by clicking "*Change*" button.

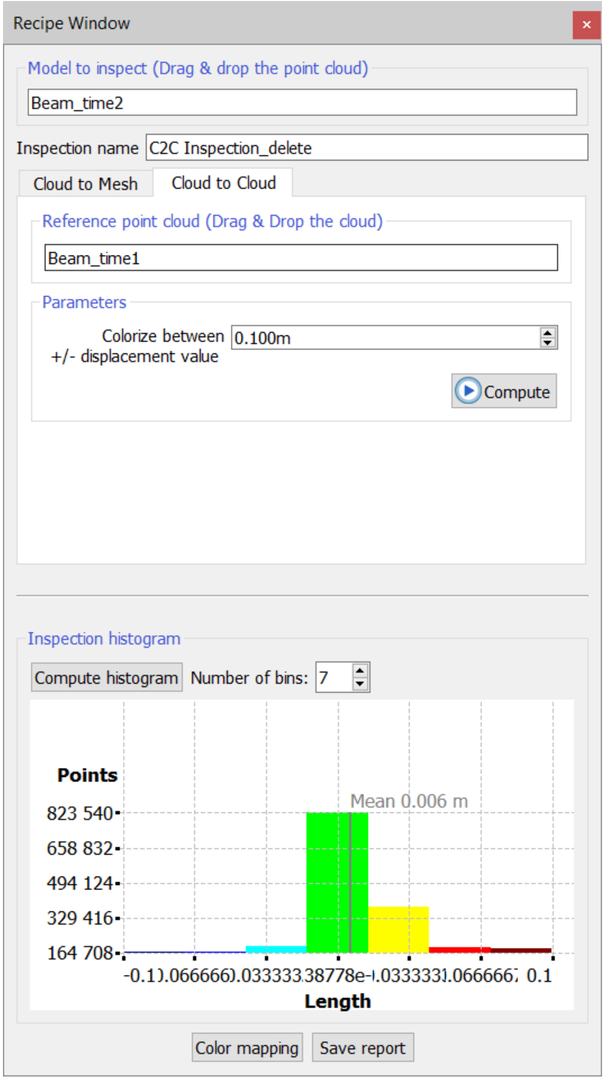


Complete inspection
The distances between inspected and reference models (meshes) are in accordance with reference meshes' normals.
Inspection along one direction

- *Cloud's origin direction*: is much faster, but on the other hand less accurate than the former two. For each point in the input cloud, is cast a ray from the cloud's origin and is computed where the ray intersects the reference surface. Finally, is calculated the distance from the point along the ray. Since this procedure only looks in one direction, it can measure infinite distance for a given point while the reference surface may be very close to the point along another direction.
- *Axis direction*: takes each point of the cloud and computes its distance from the reference model *along a specific axis*. User can choose one of the three axes of the current UCS. This inspection type is faster, is usable with a great *Max search distances*(e.g. 50m) and it provides signed distance values (signs are calculated using the normals of the reference mesh's triangles).

Distances as signed values: distances will be computed with positive sign when measure direction is oriented in accordance with the normal at the point to inspect, negative otherwise.
Distances in absolute values: distances will be computed in *absolute value absolute values*.

2. Cloud to Cloud



Input models

As inputs, user must provide a model *to inspect* and one *reference model*. Both models must be a *point cloud* (structured or unstructured).
The desired models can be inserted simply by dragging and dropping them. Press *Remove selected* to deselect undesired models from your import.

The result will be saved in a new color layer of the inspected cloud which represents the minimum distance values of each point with respect to a respective triangle built on the neighborhood of points of the reference cloud.

Parameters

- **Inspection name**: to define a name for the layer that will be generated
- **Colorize between +/- displacement value**: threshold (absolute) value used to define the limits of the scale colorization.

This inspection tool always provides a distance value for each visible point of the inspecting cloud, unlike the mesh-to-cloud inspection, where the maximum search distance constrains the maximum range of measurable distance values.

The sign of the displacement measure is conditioned by the point cloud origin; be careful if the model to be inspected is an unstructured point cloud.

After defining the inspecton modalities, press ***Compute*** to start the inspection process.
The resulting layer is appended to the color type list of the inspected point cloud, with the desired *Inspection name*.

Press **Compute histogram** button to calculate and plot (as well as to update) the histogram showing the statistics of the data distribution, after defining the number of bins you want to plot.

Use Colors mapping tool (linked in the bottom of the dialog) to optimize the colorization and the color legend.

Press ***Save report*** to collect all the information (histogram, statistics, picture with color legend, ...) in a PDF report.

Drawing tools

Constrain picking to a plane

To contrain points picked on 3D points or drawn polylines to a user defined plane.

Video Record

In Reconstructor® the creation of a video of your 3D model is possible, by using the *Video Record* commands:

Flythrough Editor [Ctrl+F]
To define and edit a trajectory that the “virtual eye” of the 3D world can go through

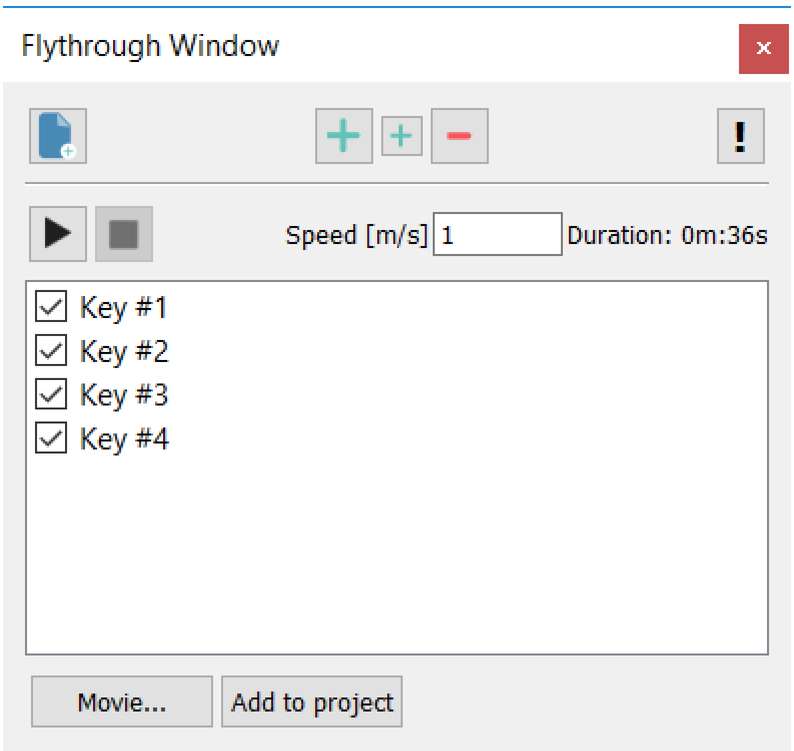
Flythrough from point list
To create a flythrough from a point list in the project

Play
To start showing a preview of your video inside the 3D window

Edit
To manage the key points and the output videos's parameters

Make movie
To extract a video

Flythrough Editor



This window allows to define and edit a trajectory that the “virtual eye” of the 3D world can go through, to generate a flythrough video. You can input a sequence of key points that you want the virtual camera to pass through, to generate impressive flythrough videos of your models.
A trajectory is made up of a sequence of view points (called Keys) which are then interpolated.

The dialog allows to

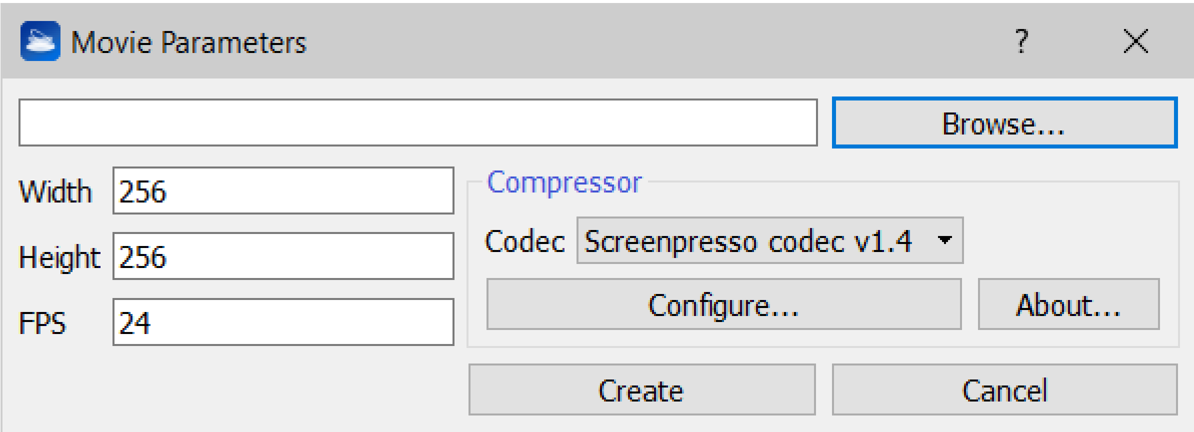
- **Add key**: appends the current view point to the list
- **Insert key before**: inserts the current view point before the selected key in the list
- **Delete key**: deletes the selected key
- **Reset list**: clears the list
- **Update trajectory**: if at least 2 keys are defined, a temporary trajectory is computed for the checked keys (uncheck to skip the key) that can be simulated (Play from) or added to

the project (Add to project)
- **Play from**: if a valid trajectory exists, an animation of the trajectory rendered, starting from the selected key, using the desired speed
- **Stop**: stops the current simulation
- **Movie**: the Movie dialog is shown to select the parameters to create a movie from the flythrough
- **Add to project**: adds the current trajectory to the current Reconstructor® project
- **Close**: hides the window

Double click with the left mouse button on a Key to jump to the view point.
If a flythrough exists in the project tree, by selecting **Edit** in its context menu the Flythrough Window is shown with the Keys that make the trajectory.

See also flythroughs and Movie dialog.

Make movie



This dialog is accessible by Flythrough Editor and it allows you to create a movie out of a video trajectory in your 3D scene.

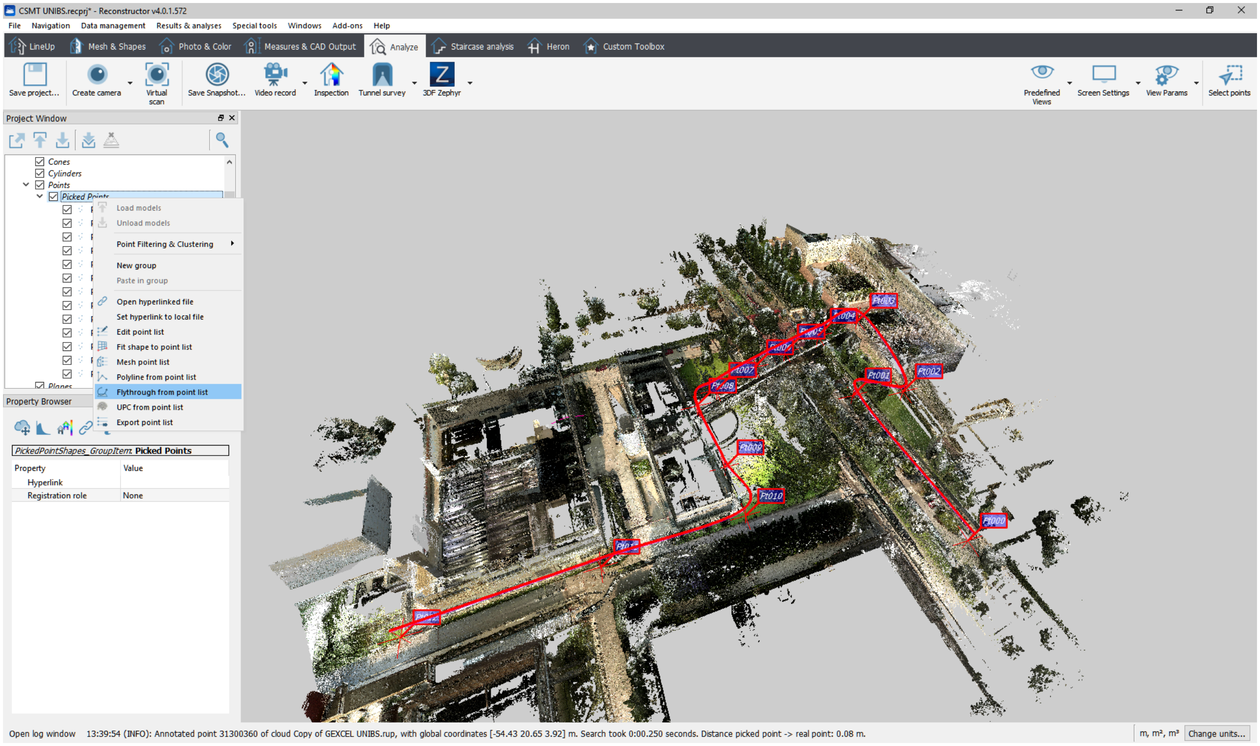
To create the video, Reconstructor® uses the video codecs founded in Windows OS for current machine, H.264 and H.265 basically.
For each encoder there is a list of presets concerning Resolution (in pixels), the Bitrate (in Mb per second) and the Framerate (in frames per second).
You can select a desired preset according to your needs and your workstation capabilities. The following presets are available: SD, HD, Full HD, 4K, including a *Custom*(izable) one. Once the dialog is closed, the Custom's parameters are saved into Reconstructor® settings and they will be available the next time you open the dialog.

After pressing *Create*, the video encoding starts and a .mp4 video is created.

See also flythroughs and Flythrough Editor.

Flythrough (trajectory) from point list

This command (contextual from point list or under *Video record* button) creates a flythrough.
It is useful to create videos and to generate orthogonal planes to it.



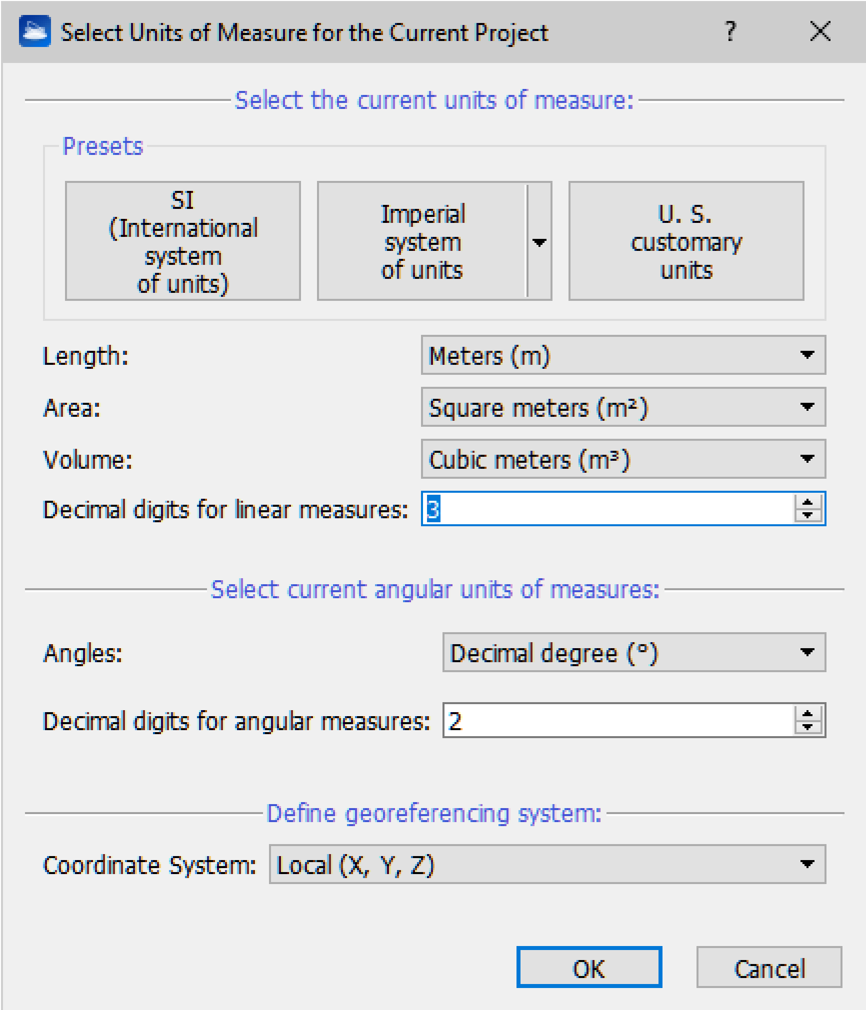
See also Edit point list, Flythrough and Flythrough editor.

Save snapshot

Save snapshot

This function allows you to capture a snapshot of the main viewport with its rendered models. You can save the captured snapshot in a variety of image formats, including Bitmap, Jpeg and PNG.

Manage units of measures



This dialog allows you to set the preferred units of measure for lengths, angles, areas and volumes; and to set how many decimal digits should appear in Reconstructor®.

All available decimal places are used for calculations.

Presets

On the top of the above dialog, you can activate some presets:

- *SI (International system of units)* will set meters as length unit, square meters as area unit and cubic meters as volume unit.
- *Imperial system of units* will set inches, square inches and cubic inches as length, area, and volume unit respectively. This button features a popup menu that gives you other two options: to use feet and to use yards as basic measure unit for length, area and volume.
- *U.S. customary units* will set the U.S. survey feet as predefined length unit, the U.S. square survey feet as area unit, and the U.S. liquid gallons as volume unit.

Selecting the units of measure

In the middle box of the dialog, you find three controls to select the length unit, the area unit, and the volume unit to be used across all Reconstructor dialogs, visualized data and outputs.
You can select among a long list of measure units, belonging to the international system of units, to the british Imperial system, and to the United States customary units, used for survey. Please see below the complete table of the units of measure supported by Reconstructor.

Selecting the data's decimal precision

The last option in the dialog allows you to select how many decimals to visualize after the comma for each output value in Reconstructor®. This gives you a lot of flexibility to format your output reports according to the precision you desire, but also it asks from you that you know the precision limitations of your sensor in the particular dataset you are working on.

Units of measure supported by Reconstructor®

Lenght units

Name	Suffix	Amount in meters
Meters	m	1
Centimeters	cm	0.01
Millimeters	mm	0.001
Kilometers	km	1000
Thousandths of an inch	th	0.0000254
Inches	in	0.0254
Feet	ft	0.3048
Yards	yd	0.9144
Chains	ch	20.1168
Furlongs	fur	201.168
U.S. links	li	0.2012
U.S. survey feet	US ft	0.30480061
U.S. rods	rd	5.02921
U.S. chains	US ch	20.11684
U.S. furlongs	US fur	201.1684

Area units

Name	Suffix	Amount in square meters
Square meters	m^2	1
Square centimeters	cm^2	0.0001
Hectares	ha	10000
Square kilometers	km^2	1000000
Square inches	in^2	0.00064516
Square feet	ft^2	0.09290304
Square yards	yd^2	0.83612736
Acres	acres	4046.8564224
U.S. square survey feet	US ft^2	0.09290341
U.S. acres	US acres	4046.873

Volume units

Name	Suffix	Amount in cubic meters
Cubic meters	m^3	1
Cubic centimeters	cm^3	0.000001
Cubic millimeters	mm^3	0.000000001
Liters	l	0.001
Cubic inches	in^3	0.000016387064
Cubic feet	ft^3	0.028316846592
Cubic yards	yd^3	0.764554857984
Gallons	gal	0.00454609
U.S. cubic feet	US ft^3	0.02831685
U.S cubic yards	US yd^3	0.764554857984
U.S. acre feet	US acre feet	1233.482
U.S. liquid gallons	US fl gal	0.003785411784
U.S. liquid barrels	US fl bbl	0.119240471196
U.S. dry gallons	US dry gal	0.004404884

Angular units

Name	Suffix	Amount in decimal degree
Decimal degree	°	1
Radians	rad	180 / Π

Special Tools

These tools are related to special environment and capabilities to interaction with other platforms.
The presence of these tools/commands is subordinated to the licensed add-ons.

Tunnel Survey

3DF Zephyr

Tunnel survey

The tunnel survey tools are used to automatically generate cross sections along a trajectory and extract a cylindrical develop of the tunnel as a scaled orthophoto.

Cylinder virtual scan
The virtual scanning can here be done in cylindrical projection using a cylindrical projector as a virtual scanner.
This function will enable you to virtually scan your 3D scene by looking at it from a cylindrical camera whose "backbone" is a selected flythrough. More concretely, Reconstructor® will split your trajectory in many segments, each segment running from point T(t) to point T(t + 1), where T(t) indicates the point on trajectory T at time t. Out of each one of those segments, a cylindrical camera will be created having as main axis the segment. Reconstructor® will pop up as many virtual scan dialogs as the segments are, to allow you to do a virtual scan.

Tunnel cross sections
To generate a series of cross sections with a defined spacing; all of them are disposed along a selected trajectory or polyline or series of points that describes the elongation of a tunnel. The sections are perpendicular to the trajectory and constrained to be parallel between them.
See more details at Tunnel cross sections page.

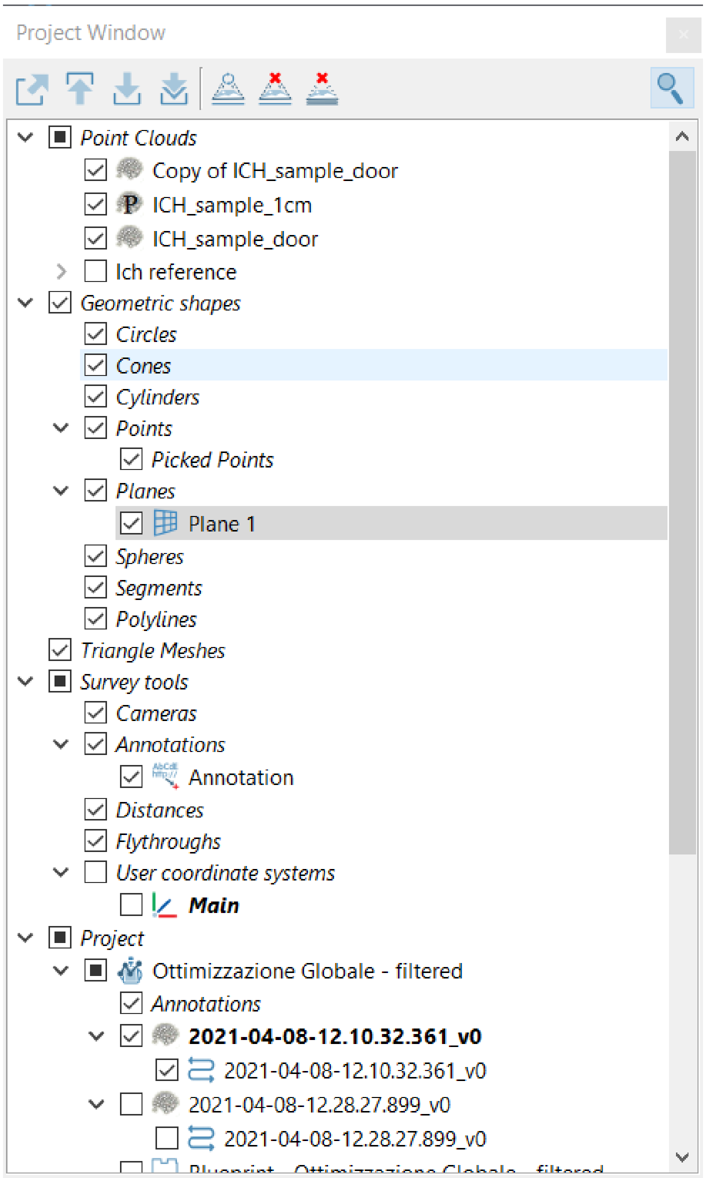
Windows menu

The Windows manu in the Menu bar shows the permanents windows in the application.
All the dockable windows can be opened by right-clicking on the Top toolbar and upon.

- *Project window*
- *Property window*
- *Readout window*
- *Point list window*
- *Log window*
- *Manual positioning (Adjust Pose)*
- *Flythrough window*
- *Recipe Window*
- *GUI options*: opens a dialog to change GUI settings.

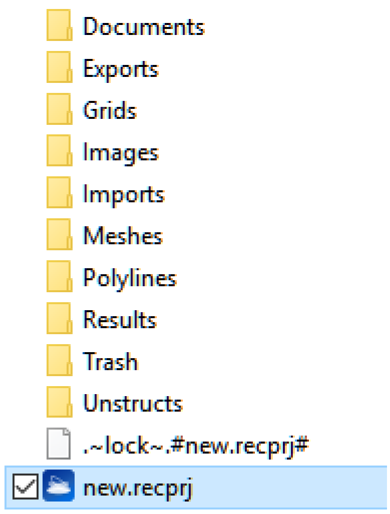
Project Window

All the item are listed in the *Project Window.*

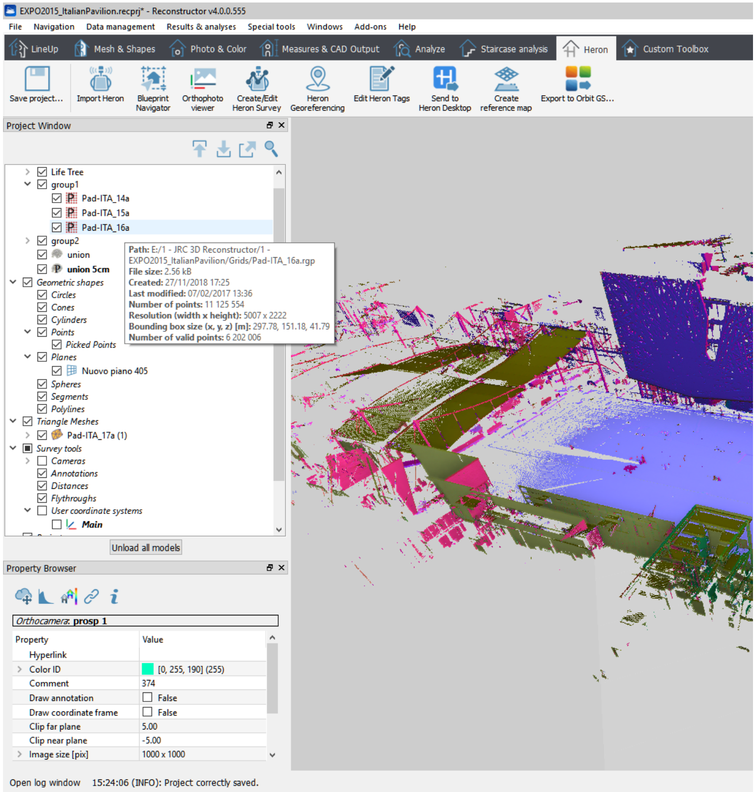


This window provides a ***tree view*** of the current project items.

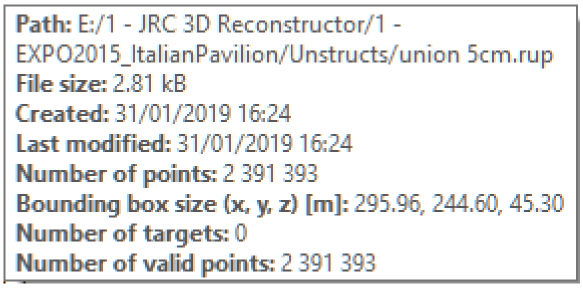
The Project window lists the items present in the project, organized in a tree structure, where each data type is grouped in a separate folder.



Note: the .~lock~.#new.recprj# is a temp file used to lock the project and prevent it from being opened in another Reconstructor session.

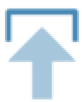


By hovering the mouse over the item name, a tool tip pop ups with some information about the item, like file path, file size, etc.



To enable the rendering of an item, click the checkbox ☑ on the left of the item. The checkbox of a folder enables/disables all its sub-items.
Uncheck-check process permits you to load a model.

Items such as point clouds or triangle meshes are *models*. Models can be *loaded* or *unloaded;* loaded models are listed in **bold** in the project window.



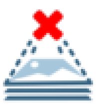
Load model
Selecting a model (cloud of points or mesh) from the project window tree and pressing this button, the model will be loaded in PC RAM and if it is also checked ☑, it will appear in the 3D viewport.

Unload model
It unloads from the RAM and from the main window the selected item(s).

Unload all models
It unloads from the RAM and from the main window all the models (point clouds and meshes).

Load projector
It loads (light on) the image of a projector on the 3D model.

Unload projector
It unloads the image of a projector from the 3D model.



Unload all projectors
It uunloads from the RAM an from the main window all the projectors.

Search project item
It searches through the items of the project.

Furthermore, by right-clicking on an item in the project window, a context menu is displayed where only the commands that apply to the given item type are shown. Commands can therefore also be issued from that context menu: this is useful in particular to load/unload a model, to set an hyperlink to an item or to open the hyperlink, to move the item to another group in the project.

Each item has properties that can be viewed and edited in the Property window when the item is highlighted.

To rename a model, right-click on it and select *rename*. To rename all other items, double click on the item's name and edit the name.
If an item needs to be saved it appears in red color or **red bold** color if loaded.

To **remove** one or more items from the current project, select the item(s) in the Project window and select *Remove* from the contextual menu or press *CANC* key. The files are not lost, they are simply moved to the project's *Trash* folder inside the project directory, so they can be recovered (cut away from the trash to an other project folder and *Load item...* from the *File* menu).
A specific behaviour is dedicated to removing groups or group's items (see more details at Groups entry).

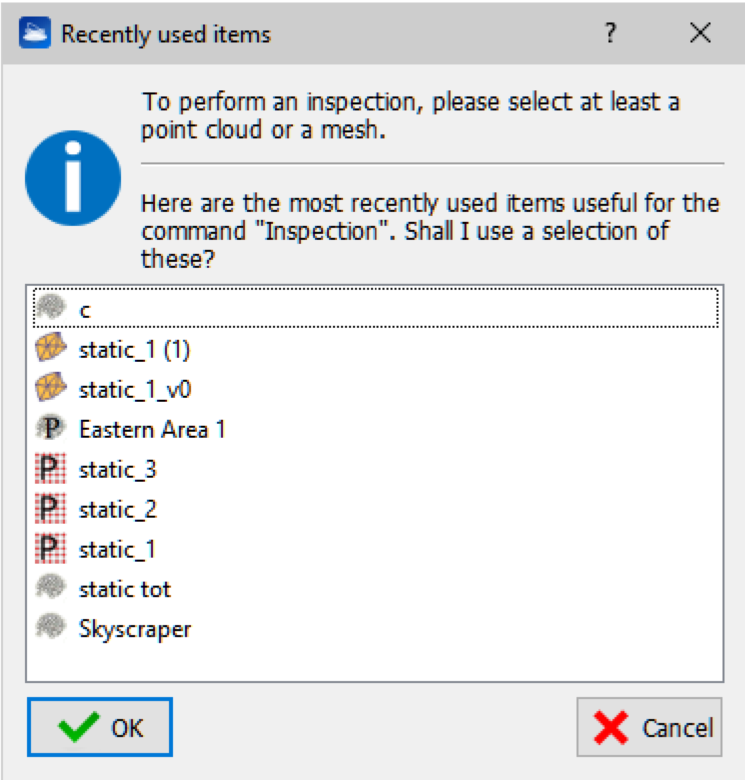
Multi-selection of items is possible by using [*Shift]* and [*Ctrl]* keys. Then it’s possible to:

- delete the selection with *CANC* key
- set the common properties in the Property window
- open the context menu (right click on the selection) to display the commands that are applicable to the selection.

Quick model distinction can be made by pressing the “Color by ID” button on the Project window toolbar. Each model is colored by its color ID set in the Properties window. This color is randomly computed when the model is inserted in the project. See Screen settings.

See also dockable windows, and project items.

Recently used items dialog



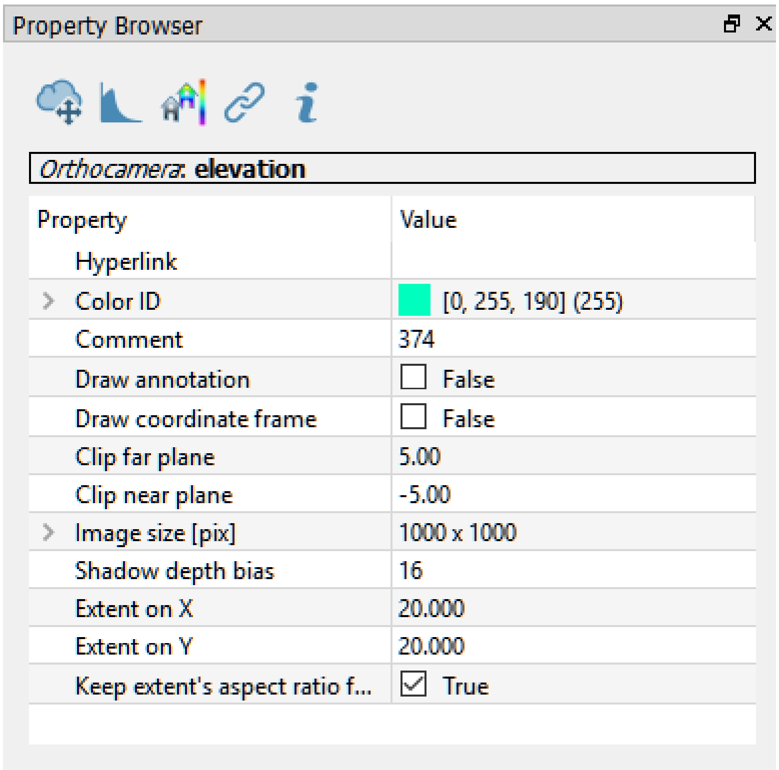
This dialog allows you to quickly select for a certain operation some of the project items you used most recently.

Many important Reconstructor® functions need to have project items as input. For example, the function “Registration” needs two grid point clouds as input. You can start these functions by taking care of selecting the input items first, in the project window. However, this dialog offers you an alternative and in some cases more efficient way of starting your functions.

If you start a function that needs items of a certain type as input, and none of these items is selected, then the dialog pictured above pops up to help you. The peculiar feature of this dialog is that it shows you the candidate items for your function, listing them *from the most recently used to the least*. If you are working repeatedly around few items, using this dialog might be more efficient than scrolling each time the project window that may easily contain hundreds of items.

See also project items, and project window.

Property Window



This dockable window is situated by default on the left side of Reconstructor®'s GUI. It allows to read and edit the *properties* of the current selected item(s) in the project window.

In Reconstructor® each project item has *properties*.
There are *general* properties, owned by all project items, and there are *specific* properties that are peculiar to each type of project items.

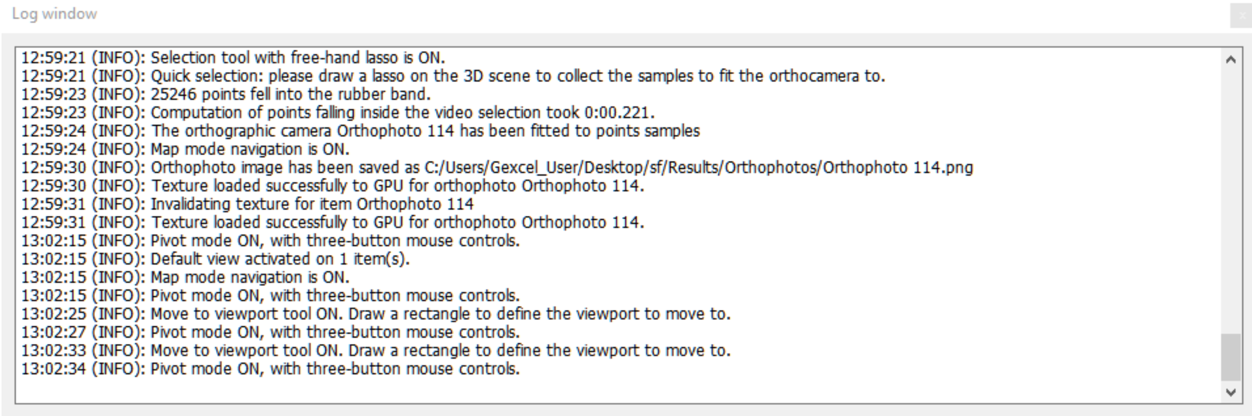
General properties

These properties are common to all Reconstructor® project items, and always displayed in the property editor:

- *Hyperlink*: each project item can be linked to any URL: other project items, files on the PC, web addresses.
- *Color ID*: each project item has a specific color to identify it from other items. You can edit the color ID by clicking on the *value* field of the color ID property. To render the items with their color IDs, select *Navigation->Color by ID.*
- *Comment*: this comment will appear in the item's annotation if the above property is true.
- *Draw annotation*: a boolean property. Each project item has an *annotation*, that is rendered in the 3D scene if this property is true.
- *Draw coordinate frame*: this property is not owned by annotations. All other project items have an *object coordinate system*. If this boolean property is true, Reconstructor will render an axes triplet to show where the object coordinate system is. This is useful for examples to know in which direction a plane's normal is oriented.

Log Window

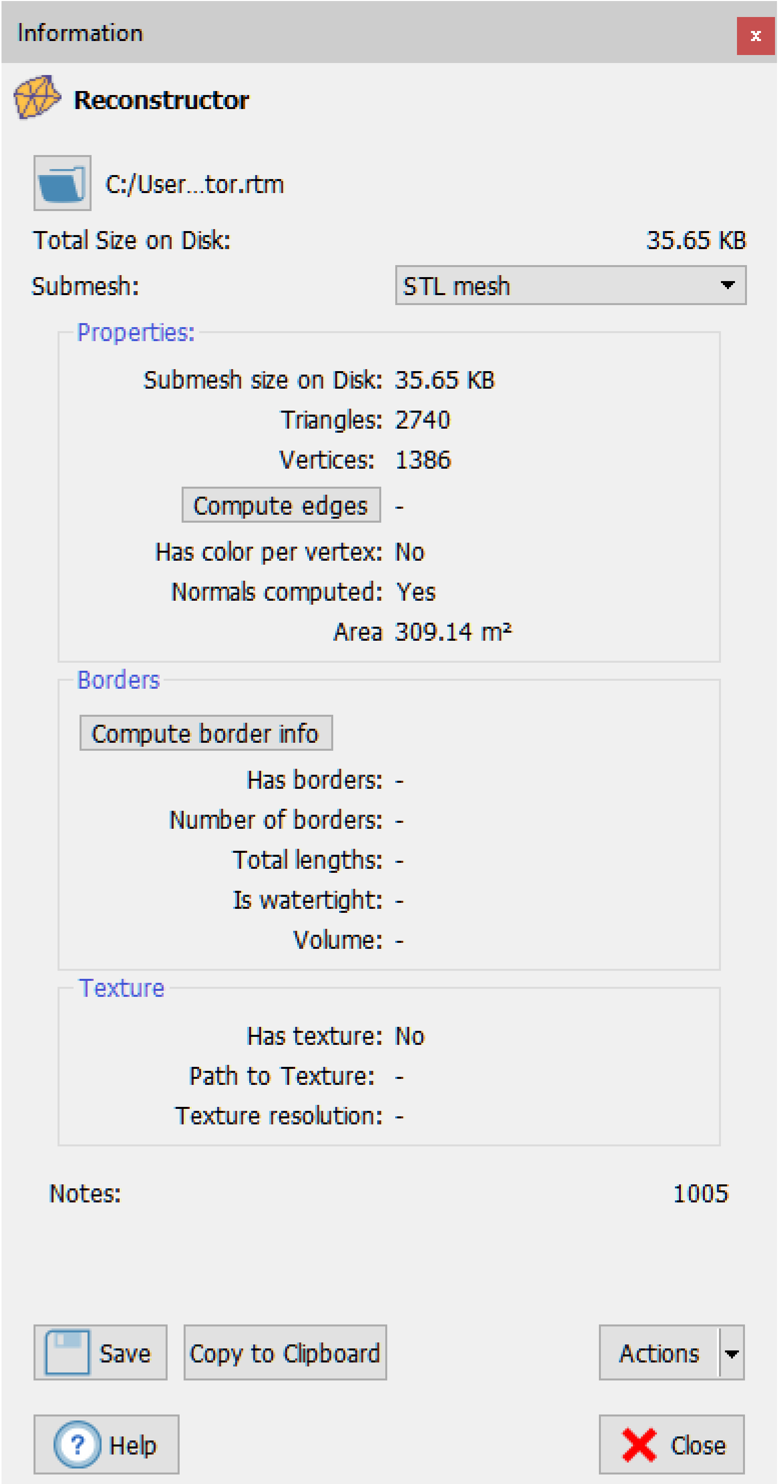
By pressing the *Open Log Window* button, at the bottom left screen corner, you will see the last twenty lines of the log.



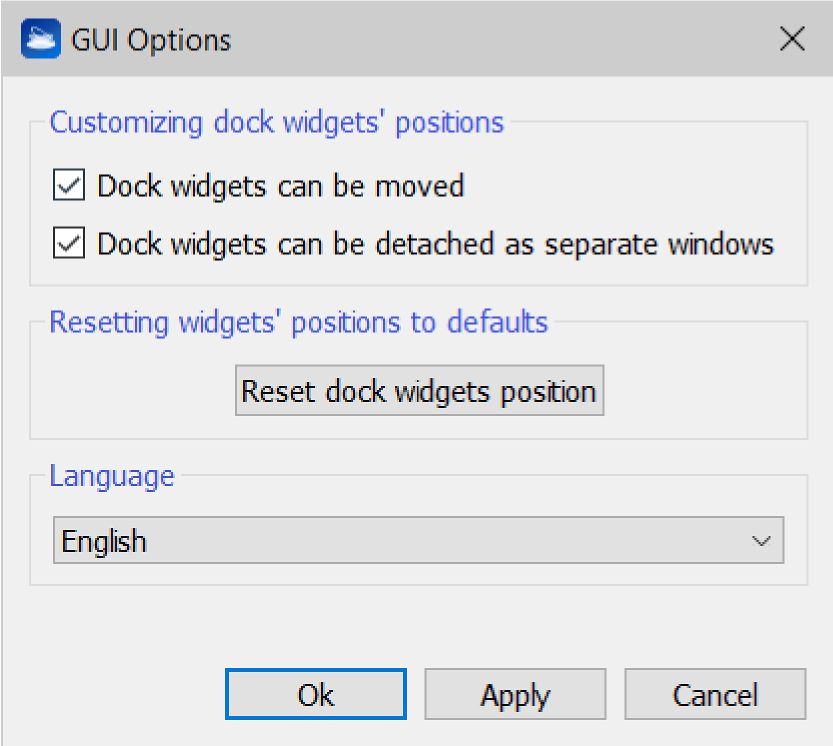
Button clicks, status of algorithms, errors and warnings, result of procedures... everything that happens inside Reconstructor® is logged and shown in the text panel displayed in this window. Log messages can be of four types: information, warning, critical and fatal. Log messages appear always with their type and the time instant.

Information window

The information dialog includes general properties, points statistics, color and geometrical properties of the Reconstructor's items.

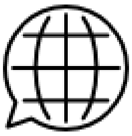


GUI Options dialog



This dialog allows the user to change some GUI settings.

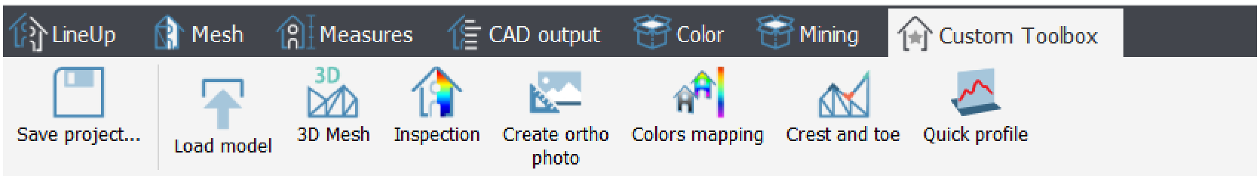
The first two check boxes control how freely the dockable widgets can be moved around and detached.
A reset button resets widgets' positions to a default setting.



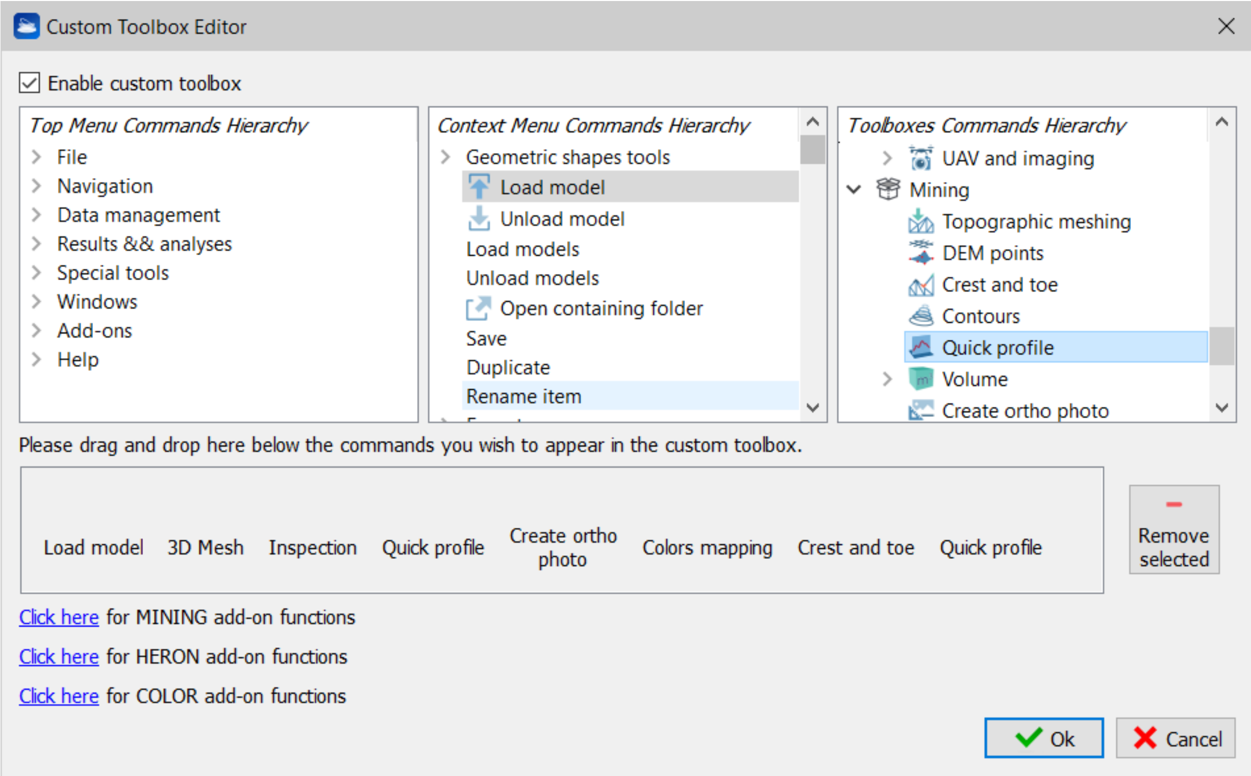
The software's language is managed by a drop-down menu.
After selecting on of the available languages (English, Italian, Russian and Chinese), click on Apply.
Restart Reconstructor® for the language update to take effect.

Custom toolbox

In the Toolbox bar all the main tools are organized in different tabs of default, as you seen in the previous paragraphs.
If you are used to using so me commands often, it can be useful to create a customized bar that includes these commands, available under "*Custom Toolbox*".
As for example...



The Windows > *Custom toolbox* tool opens a dialog to define the toolbar, available also by clicking on  in the Toolbox bar.



Three main boxes include all the tools available in Reconstructor®, subdivided into *Top Menu Commands*, *Context Menu Commands* and *Toolboxes Commands*, according to their position in the software.

To add one of these commands to the Custom toolbox you need to select and drag&drop it to the area at the bottom of the dialog. You can also double-click on any command to add it to the Custom toolbox.
To remove one of the commands from the Custom toolbox select it in the area at the bottom of the dialog and click on the *Remove selected* button.

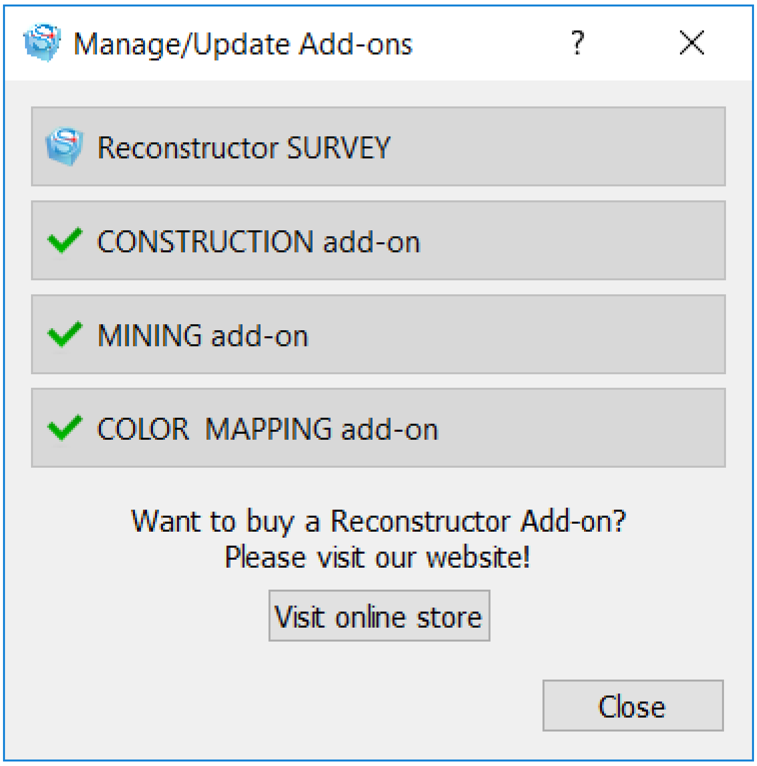
By flagging *Enable custom toolbox* it will be available on the Toolbox bar.
Click on *Ok* to confirm and close the dialog.

Add-ons

Reconstructor includes all the functionalities to import, process, automatic aligned and measure 3D clouds. It's easy to add the Mining add-on for mining and terrains application or the Color add-on to integrate 3D models with external images.

Full functions in trial version: during the trial period you can test all the add-on and verify what you need.

By clicking on *Add-ons* > *Manage Add-ons* command a selectable list of the add-ons will appear, giving you the possibility to activate or deactivate the add-ons.



Note: the access of each add-on is due to the available licenses!

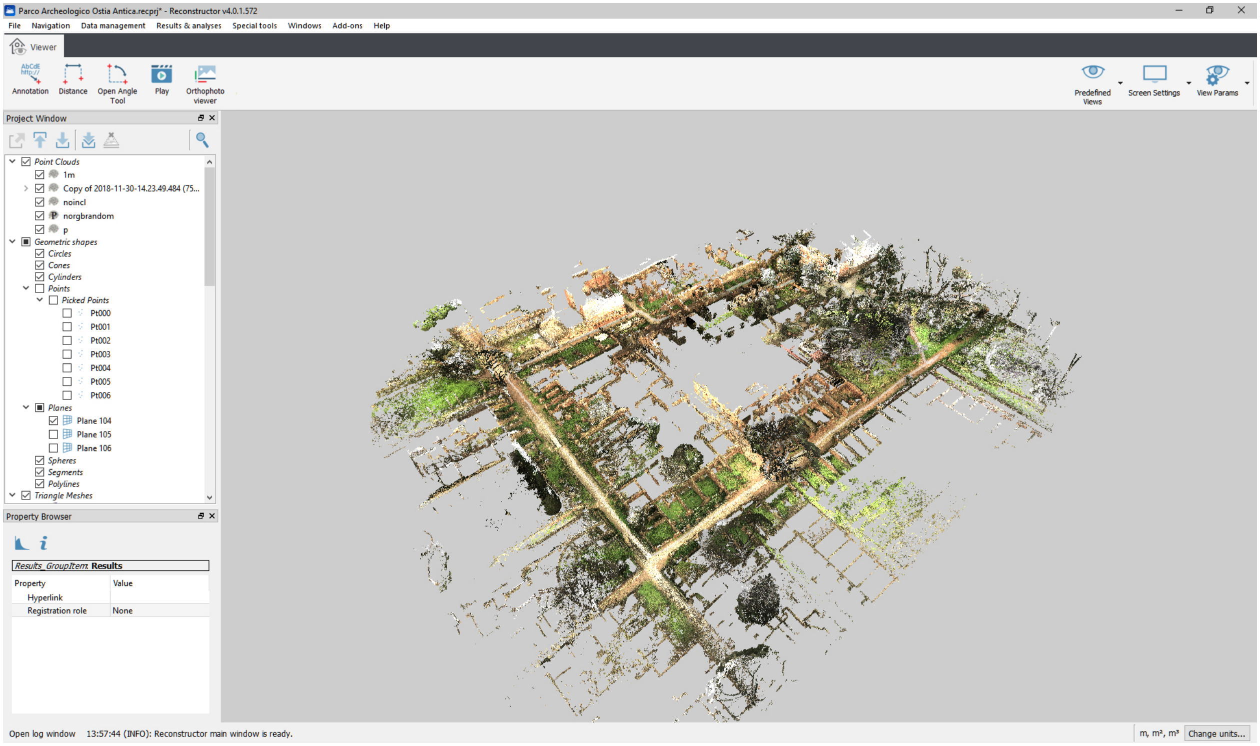
As an example, if an user has to carry out a task with mining data, he or she can adjust the interface for a Mining workflow, through this menu. In this way, top toolbar and dockable windows are reorganized by showing only the functions useful for mining tasks, and in a meaningful order and place.

If you uncheck all the buttons, Reconstructor runs in Viewer mode.

- Reconstructor Viewer
- **CONSTRUCTION add-on**
- **MINING add-on**
- **COLOR MAPPING add-on**

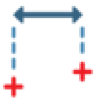
Reconstructor (free) Viewer

The Reconstructor Viewer's toolbar has only a few available functions.
All Reconstructor navigation and display functions are included.



Annotation

This function allows you to create annotation of 3D points of models in your project.



Distance

This button allows you to measure a distance between any two points in the 3D scene

Open Angle Tool

This button enables you to measure and angle between three points in your 3D scene.

Play

Plays an animated flythrough of the 3D scene, according to a given trajectory.

Orthophoto Viewer

To open a 2D viewer to visualize and dimension 2D maps created in Reconstructor (both x-ray orthophotos and standard orthophotos).

CONSTRUCTION add-on

Dedicated functions for BIM, Architecture and Construction.

MINING add-on

Dedicated functions for territorial data analysis.

Expand your Reconstructor commands with the MINING add-on functions: advanced calculation of volume/cut&fill volume, topographic mesh, extraction of sections from plans and much more. This package of functions is particularly suitable for those working on spatial data, quarries or open pit mines to optimize the development of a site and make work progress decisions based on detailed information.

COLOR MAPPING add-on

Dedicated features for images management.

Expand your Reconstructor commands with the COLOR MAPPING add-on features. You will get at your disposal the creation of meshes with photographic textures, the use of panoramic images, the management of the projectors and much more. This set of functions is particularly suitable for those who want to work with high-resolution images to integrate with their 3D models and obtain high-level results such as technical tables with photographic details.

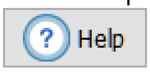
Help

Through this menu you can access Reconstructor® Online Help system, diagnostic information, useful when asking for support.
You can also enter Reconstructor® license manger, to browse the license features activated and activate new ones.

In detail see links down here:

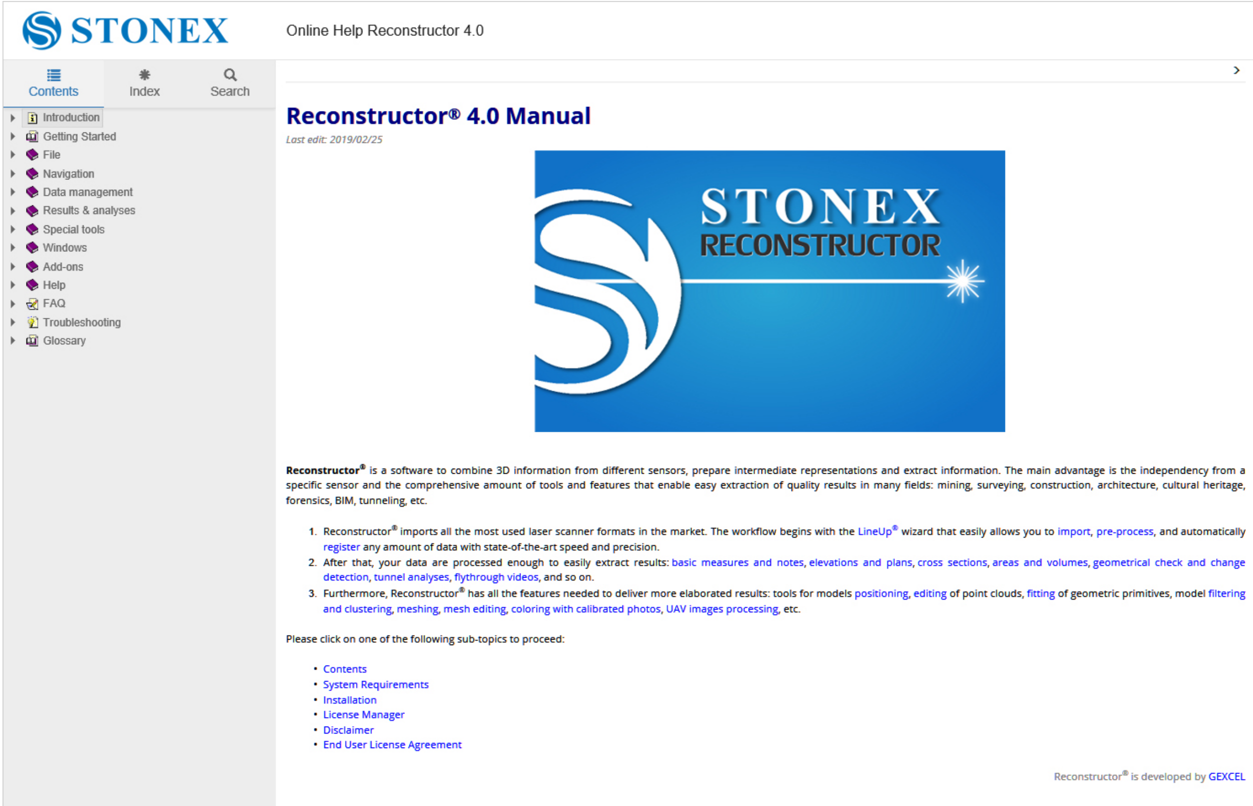
- Online Help
- Offline PDF Manual
- License Manager
- Show Commands Dashboard
- Memory Usage Report
- Hardware Info
- About Reconstructor
- Check For Updates
- Open Log Folder
- Crash report

Online Help

This menu voice opens the present online help (which can be invoked also by pressing F1 key).
You can open the contextual online help about the current window by just pressing F1 if the button 

is present.

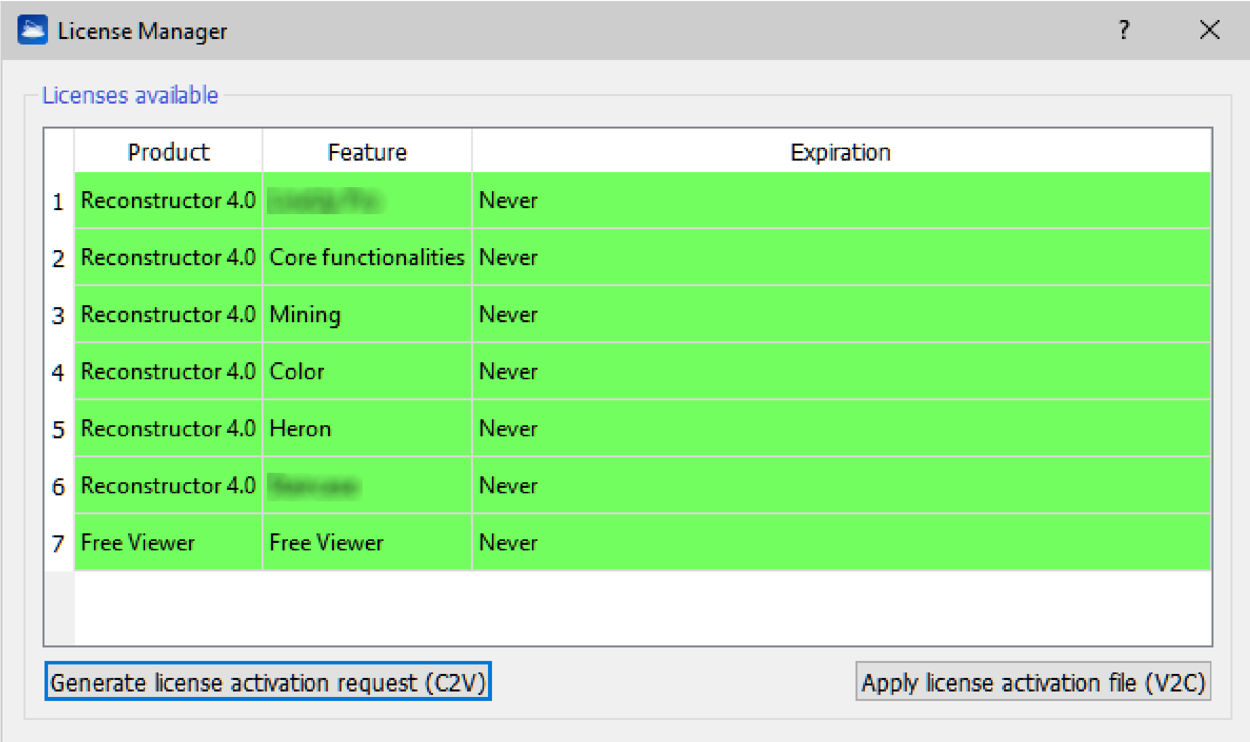
The last update date is written in the Introduction page; this online help is more frequently updated than Offline PDF Manual.



Offline PDF Manual

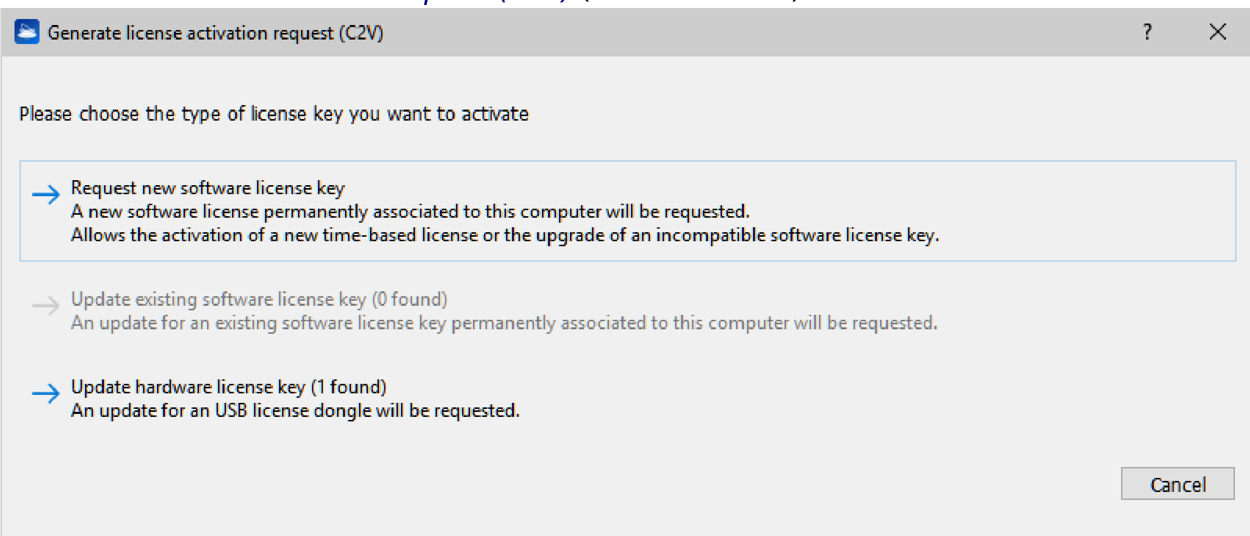
This manual is updated when a new version of Reconstructor is installed. It doesn't need an Internet connection.
We recommend using Online Help because it is more frequently updated and better formatted.
Use it only if you do not have an Internet connection available.

License Manager



Here you can check your current license or activate / update a new one.

Generate license activation request (C2V) (Client to Vendor)



The software checks your licenses and offers you only the possible choices.

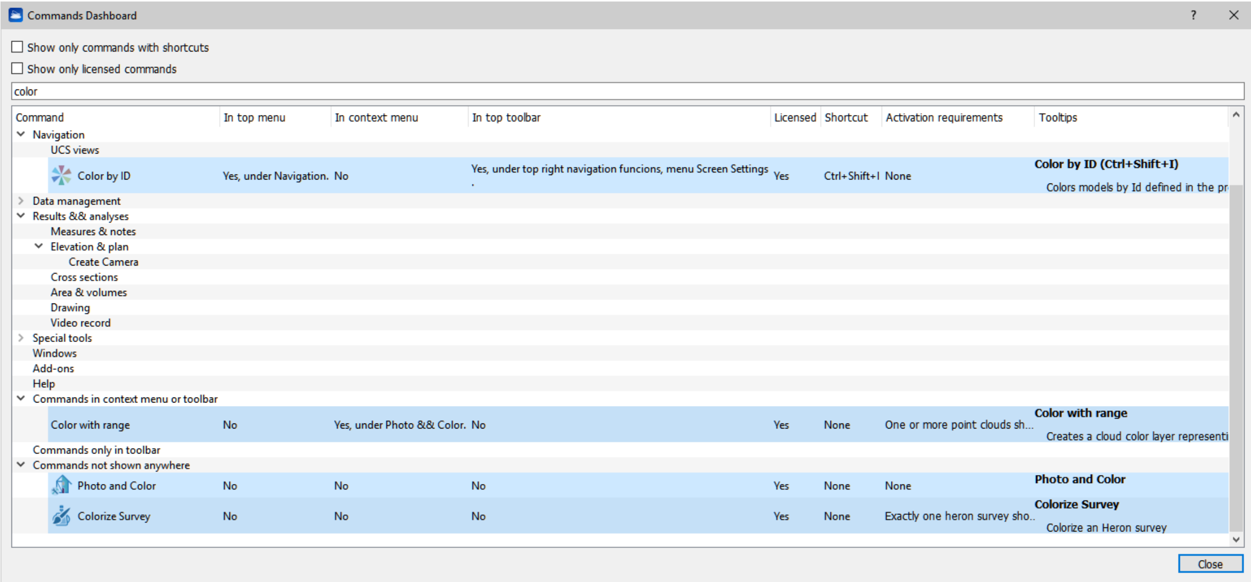
Apply license activation file (V2C) (Vendor to Client)
Browse for the *.v2c file.

See Licensing chapter for license details, request and activation.

Commands Dashboard

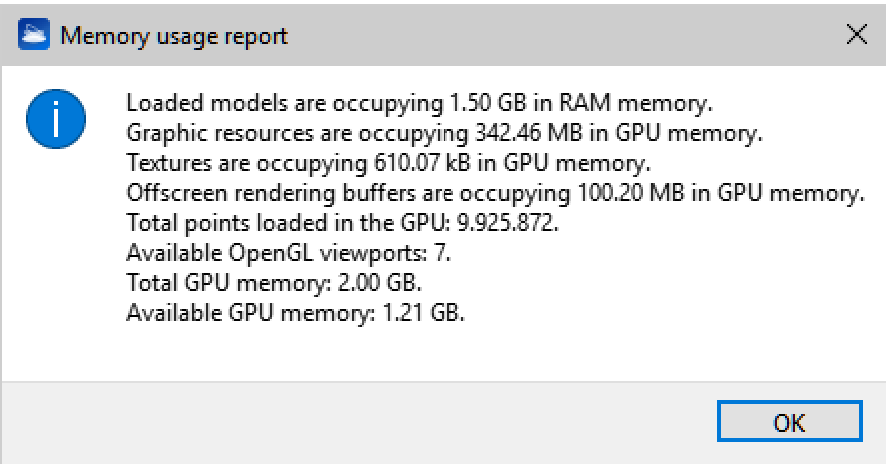
The *Command Dashboard* (*Ctrl+d*) supports the user to find the location and the meaning of all the

Reconstructor commands.
It shows also the commands short cuts when available.
There is a very useful search bar, use it!



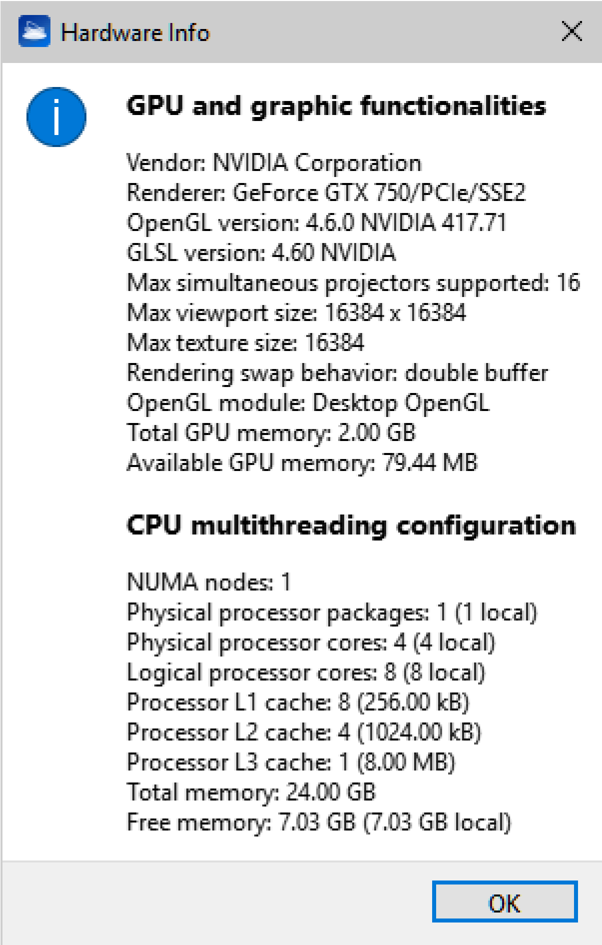
Return to the Getting Started where the main commands are briefly explained.

Memory Usage Report

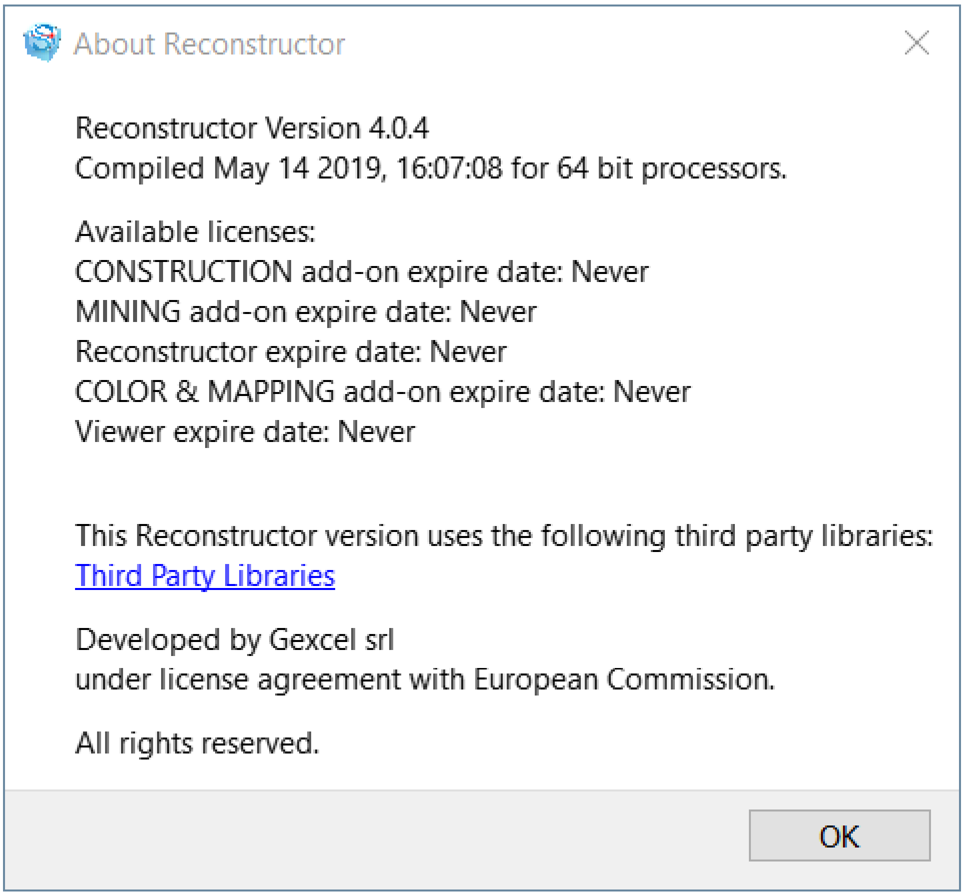


Some information about the status of memory (RAM and GPU memory).

Hardware info



About Reconstructor®



This menu option opens the *About Reconstructor®* window. This window displays Reconstructor®'s complete version number and other diagnostic information. When asking for support from Stonex, you may be asked to produce a screenshot of this dialog, or to report some information from this window.

This Reconstructor version uses the following third party libraries.

Third Party Libraries

Library	link
assimp	http://www.assimp.org/index.php/license
ceres-solver	http://ceres-solver.org/license.html
eigen3	http://eigen.tuxfamily.org/index.php?title=Main_Page#License
IfcOpenShell	http://www.ifcopenshell.org/
lib3ds	https://github.com/hoopoe/lib3ds
libcurl	https://curl.haxx.se/docs/copyright.html

libraw	https://www.libraw.org/docs
loki	http://loki-lib.sourceforge.net/index.php?n=Main.License
marble	https://marble.kde.org/index.php
OpenCASCADE	https://www.opencascade.com/content/licensing
opencv	https://opencv.org/license.html
OpenMesh	https://www.openmesh.org/license/
openvrml	http://www.openvrml.org/doc/lgpl.html
Poisson Reconstruction	http://www.cs.jhu.edu/~misha/Code/PoissonRecon/Version 10.07/
proj4	https://proj4.org/about.html#license
QuaZip	https://sourceforge.net/projects/quazip/
Qwt	https://qwt.sourceforge.io/qwtlicense.html
Snappy	https://github.com/google/snappy/blob/master/COPYING
Xerces	https://xerces.apache.org/xerces-c/
LASlib	LASlib

LASlib

LICENSE AGREEMENT:

This software is distributed WITHOUT ANY WARRANTY and without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.

There are TWO parts to LAStools. One part is OPEN source (LGPL 2.1). The other part is CLOSED source and requires licensing for most commercial or government use. If you're unsure please email 'martin@rapidlasso.com' before using LAStools. For education and evaluation purposes you can use and test LAStools as much as you need to ... (-:

Personal note: The creator of LAStools and LASzip is currently not able to attend LiDAR conferences or other scientific meetings in the US over

a somewhat unfortunate, kind of silly, and in many ways absurdly funny
incident involving "laser chickens" and a US Nuclear Weapons Lab. If you
have enough "pull" to rectify this ridiculous situation we would like to
hear from you ... (-;

This part of LAStools is "free" open-source LGPL (see COPYING.txt). To
purchase a different licensing model contact 'license@rapidlasso.com'.

* laszip
* lasindex
* lasvalidate
* lasliberate
* lasinfo
* las2las
* lasdiff
* lasmerge
* las2txt & txt2las
* lasprecision
* LASzip (with static linking exception)
* LASlib (with LASzip) - the API used by LAStools.

These two are also "free" to use but not available in open source.

* lasview
* laspublish
* lasoptimize

The following LAStools require licensing for any commercial, government,
or production use. However, they may be used "freely" for all non-profit
personal, non-military educational, or non-profit humanitarian purposes.
Note that the output of the unlicensed version can be slightly distorted
after certain point limits are exceeded. Control output in the console
(aka "the black window") informs the user whenever this happens.

* blast2dem
* blast2iso

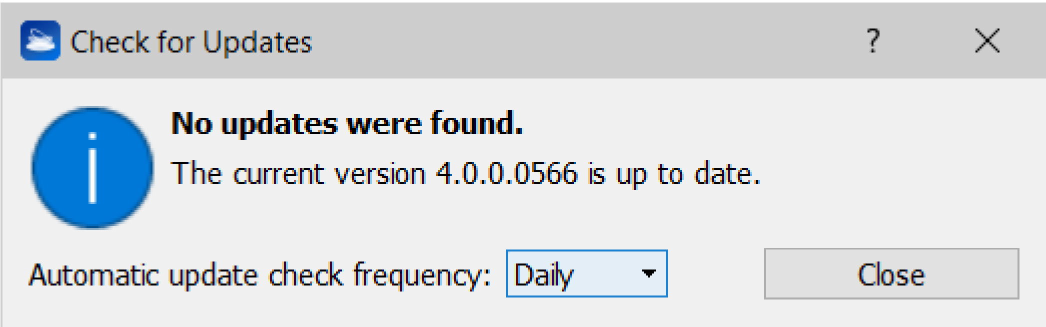
* lasground & lasground_new
* lasheight & lastrack
* lasclassify
* lasgrid & lascanopy
* lasboundary
* lascontrol
* lasoverlap
* lasoverage
* lasduplicate
* lassplit
* lasreturn
* las2tin
* las2iso
* las2dem
* lasthin & lasnoise
* lassort
* lastile
* lasplanes
* lascolor
* lasclip
* las2shp & shp2las.

If you want to use LAStools please contact us for licensing. You can

(a) license the existing executables for production
(b) license the existing executables for inclusion in your software

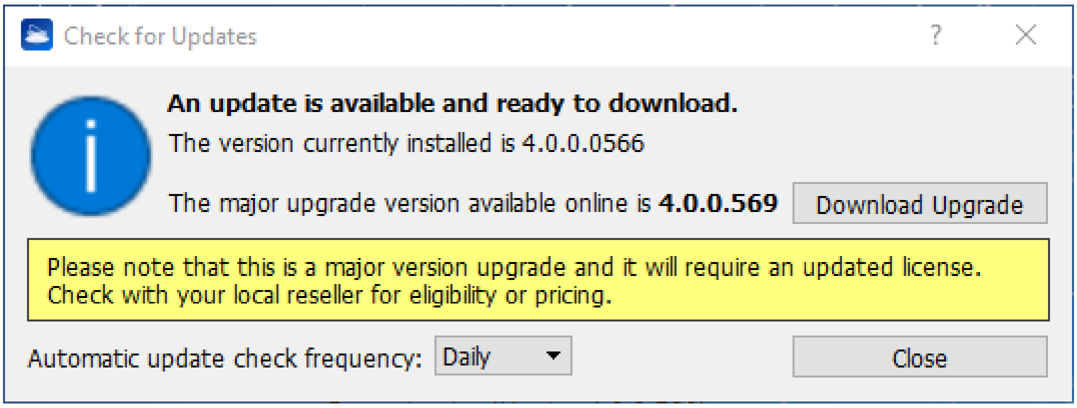
You can also contract with rapidlasso GmbH to create LAStools tailored to your particular needs.

Check for Updates



Reconstructor® automatically checks new updates when available.

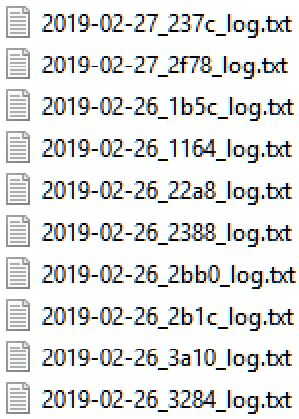
It is also possible to set an automatic check of update frequency: daily, weekly and monthly.



After downloading, follow the instructions to update the version.

Open Log Folder

This button opens the Gexcel log folders at the address *C:\Users\<username>\AppData\Roaming\Gexcel\Reconstructor* .



Crash report

In the event of a crash, the Reconstructor® application closes and a dialog is shown that invites to send a crash report and (optionally) allows you to send a feedback message to attach to the report.

Optionally providing your email address you can be contacted to directly support you. you accept that your data will be internally managed by Stonex and only used by Stonex to provide support and to improve the performances of Stonex's software and products. Your email address will be accessible to Stonex's Technical and Administration teams and will not be used for commercial or marketing actions.

The crash report dialog then allows you to restart the application or exit.

Glossary

In this section you can find all the main definitions that you can encounter during the usage of Reconstructor®.

Annotations

An *annotation* is a project item that defines a 3D point annotation complemented by different properties below explained.

- *Annotation ID*: code to exactly identify the item in the project
- *Annotation Name*
- *X, Y, Z* coordinates of the annotation position in the current UCS
- *Color mapped layer*: current mapped color layer name
- *Value*: value of the color mapped layer (one value for scalar color layers, 3 RGB components for not scalar layers)
- *Comment*: notes at the point
- *Hyperlink*: hyperlink set to a local files or another project's item (interesting feature to add meta-information to your 3D scene from e.g. the web)
- *Color ID*: color ID of the item annotation

Annotations are stored in form of point collections. Therefore, they don't have a pose matrix like all the other project entities and cannot be moved or rotated through Manual positioning or the Pose dialog.

Annotations can be converted to *Points* or *Targets* items through the contextual menu, after selecting them in the Project Window. It is also possible to do the reverse.

Learn how to insert an annotation at Annotation and how to export it at Export annotations.

Cameras

A *camera* is a project entity that defines a viewpoint in your virtual 3D world. A camera has a *pose* (like all project entities) that defines the camera's position and orientation. A camera has also a *projection* that defines how the camera sees the world.
There are four types of cameras, according to their different projections:

- Perspective cameras
- Orthographic cameras
- Cylindrical cameras
- Spherical cameras

The editable properties of the cameras are:
- *Comment*: user comment
- *Draw coordinate frame*: draw the local coordinate frame of the frustum
- *Clip far plane*: distance of far clipping plane in meters. Specifies the maximum rendered

depth. Must be greater than Min depth

- *Clip near plane*: distance of near clipping plane in meters. Specifies the minimum rendered depth. For perspective cameras, only values greater than zero are allowed
- *Image size (Width, Height)*: size of the image in pixels (used only as a place holder for repetitive virtual scanning)
- *Shadow depth bias*: This parameter concerns shadows calculation for a projector. Sometimes, during shadow calculation, there can be some noise while assessing which object is in foreground and which is in shadow. If you increase this parameter, the effect of noise is reduced, however the shadows risk to be not realistic anymore. If you decrease it, you will get more precise shadows, but you risk to get *salt and pepper* effects: as if the color projected fell also on the object in shadow in small grains.

Furthermore, there are specific properties for each of the camera projection types:

- Perspective cameras:
  - *Vertical field of view (degrees)*
- Orthographic cameras:
  - Extent on X [m]
  - Extent on Y [m]
  - Keep aspect ratio for image size
- Cylindrical cameras:
  - *Length [m]*
  - *Radius [m]*
  - *Longitude begin [deg]*
  - *Longitude end [deg]*
  - *Slices*
- Spherical cameras:
  - Radius [m]
  - Longitude begin [deg]
  - Longitude end [deg]
  - Latitude begin [deg]
  - Latitude end [deg]
  - Slices
  - Stacks

These particular properties, that change with the camera type, define the ***frustum*** of the camera, together with the near and far plane. The *frustum* of a camera is the 3D region of space that the camera sees. It is a pyramid for perspective cameras, a 3D rectangle for orthocameras, a cylinder and a sphere respectively for a cylindrical and spherical camera.

From the contextual menu of any camera, you can activate also the following commands:

- **Go to**: to jump to the camera view point.
- **Invert direction**: to invert the direction of the view of the camera
- **Apply projection**: to jump to the camera view point and apply the camera projection as the current rendering projection (if perspective or orthographic)
- **Duplicate as**: to create a new camera copying the transformation of the current one
- **Fitting**
  - **Set optimal depth range**: to automatically compute the tightest near and far clip plane positions
  - **Select projectors in this frustum** controls which cameras meet these two conditions:

    a) have a focal axis that forms an angle with the focal axis of the selected chamber lower than "Maximum incident angle"

b) have a frustum that intersects the frustum of the selected camera

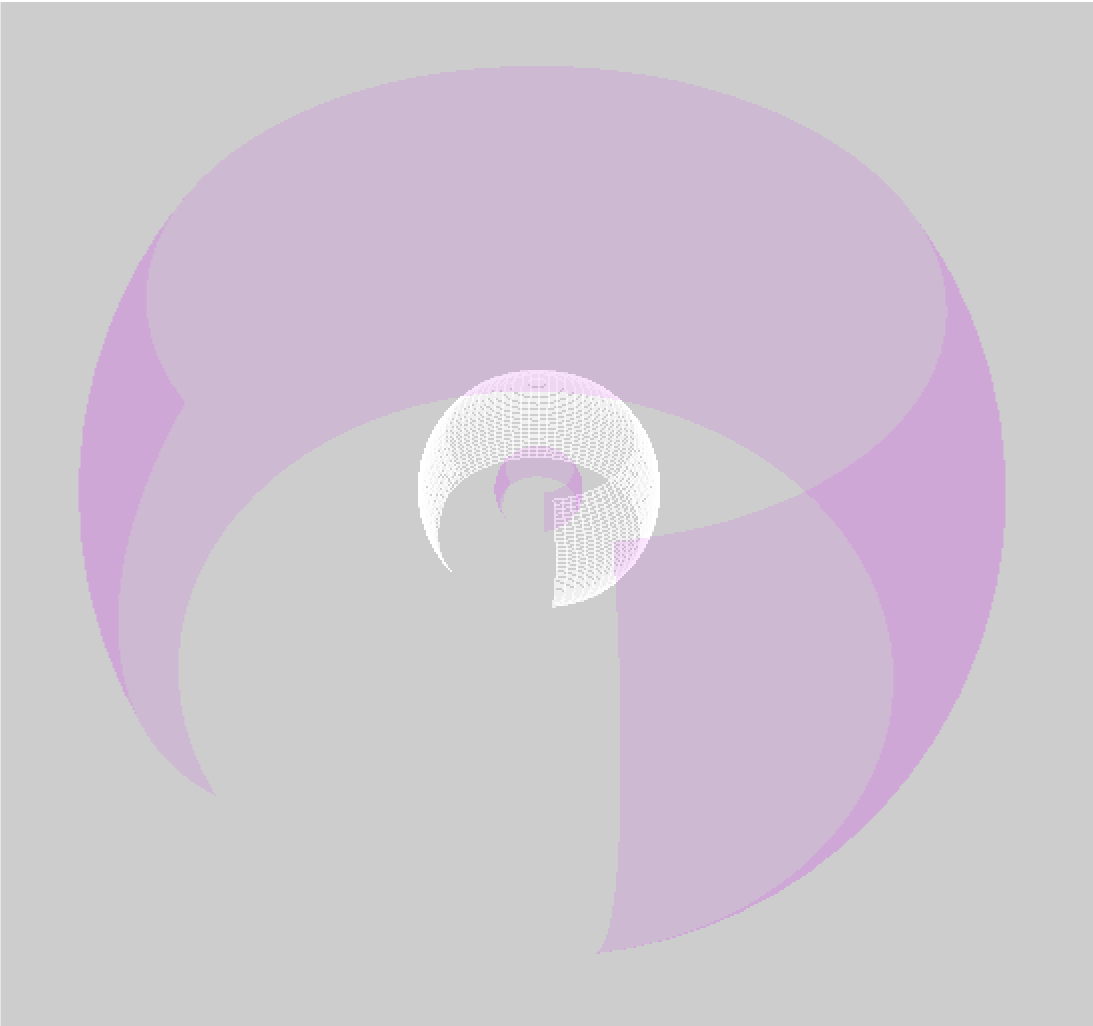
The projectors/cameras are so checked in the project window

- o **Select models in this frustum** finds (and checks) the meshes intersecting the selected camera (useful for texture mapping)

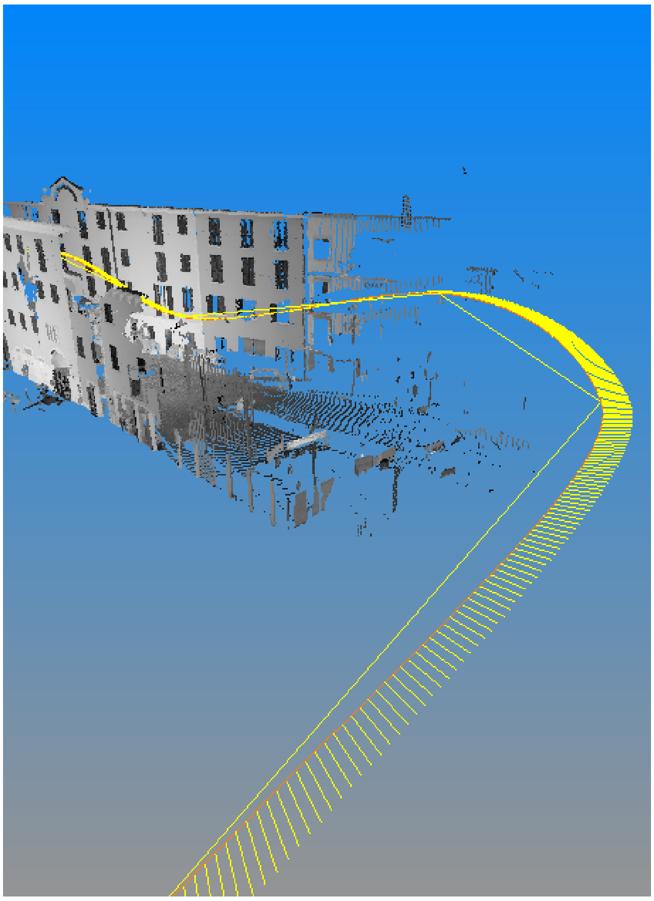
- **Analyze -> Virtual scan**: uses the camera frustum to resample the scene and generate a new grid point cloud or an image.

***Warning*: to optimize the depth accuracy of the virtual scanner, try to keep the clip near and far planes as close as possible to the desired scene depth range.**

Example of spherical camera frustums:



Flythrough (video trajectory)



A *flythrough* , or *video trajectory*, is a trajectory that runs in your 3D scene, defining how a video camera will move around in your scene to capture a movie. A flythrough is the “backbone” of the movie you're creating of your 3D scene.
In the picture above, the red curve line is the trajectory that the video camera does. The yellow segments are the optical axes of the video camera for each video frame.

Flythrough creation

The easiest way to create a flythrough is by using the flythrough editor: you can define keys, or important positions you want your video to pass through, and then interpolate the remaining frames. You can also create a flythrough from point list. The difference is that with the point list window you can only use points that belong to existing models to define your trajectory, meanwhile with the flythrough editor you can choose your keys freely.

Operations on flythroughs

If you right-click on any flythrough, you have access to some useful operations. You can play the flythrough, to see a preview of the video associated with it. You can edit the flythrough, and then the flythrough editor will appear to allow you to add or delete keys to your trajectory. You can make movie, and then the movie dialog will enable you to specify the video encoding parameters to produce the final video.

- *Cylinder virtual scan.* This function will enable you to virtually scan your 3D scene by looking at it from a cylindrical camera whose “backbone” is the flythrough. More concretely, Reconstructor® will split your trajectory in many segments, each segment running from point T(t) to point T(t + 1), where T(t) indicates the point on trajectory T at time t. Out of each one of those segments, a cylindrical camera will be created having as main axis the segment. Reconstructor® will pop up as many virtual scan dialogs as the segments are, to allow you to do a virtual scan.

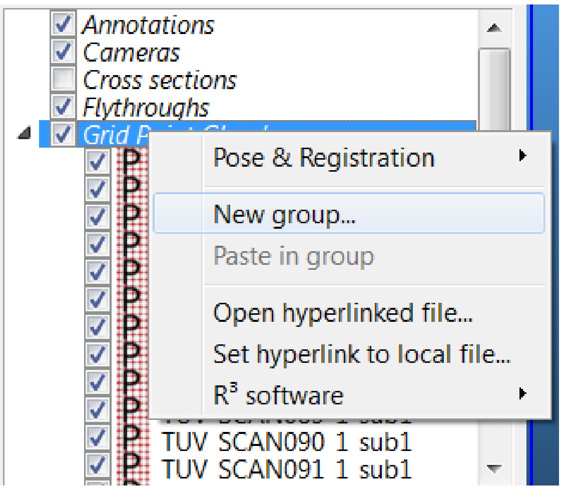
- *Generate cross sections.* This function provides you with a way to create cross sections of your models along the trajectory. If you select this function, Reconstructor® asks you the spacing between different cross sections along the trajectory. When the spacing is set, Reconstructor®

creates several planes, with the normal placed along the trajectory and placed according to the spacing selected. These planes are added to the project and named *Flythrough section n(N)*. It is left to you to actually select one of these planes and calculate the desired cross section.

You can also convert the flythroughs into polylines by using the *Convert to polyline* command in the contextual menu.

Groups

A *group* is a cluster of items composing the project.
You can create and manage the groups from the context menu.



To *create* a group you only have to open the context menu of a data type folder by right-clicking on it and select the ***New group...*** command.

To populate a group you have:

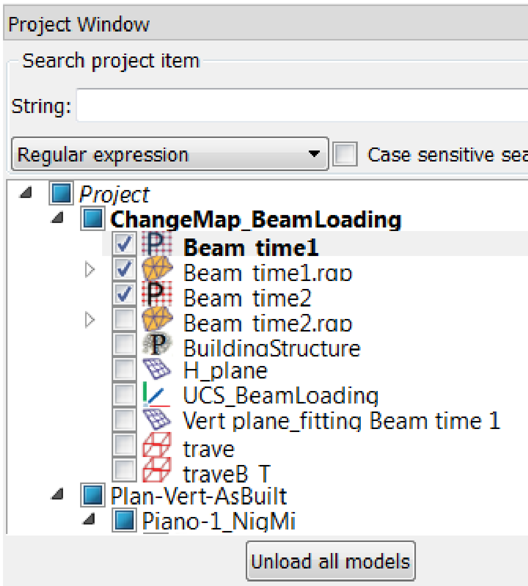
- to move a selection of items from a previous position (to a new one) with ***Move*** command from the context menu of the selection
- to paste the selection of items in the new group, with ***Paste in group*** command from the context menu of the new group.

A potential hyperlink is visible in the Property Browser of a group.
Here is also possible to set a Registration role for all the scans inside the group:

- *None*
- *Same as parent*: to set as registration role the same role of a higher level group
- *Children move together*: if a single scan is moved in the 3D space, all the other scans in the group move together, as a rigid system

These roles are used to make a scans registration through groups.

You can also create a group made up of different data type objects. As is seen in the picture at its side this possibility is useful to put together all the items of a single sub-project, for example.



To move out by a group an object, the ***Ungroup*** command from the context menu of the object is the way.

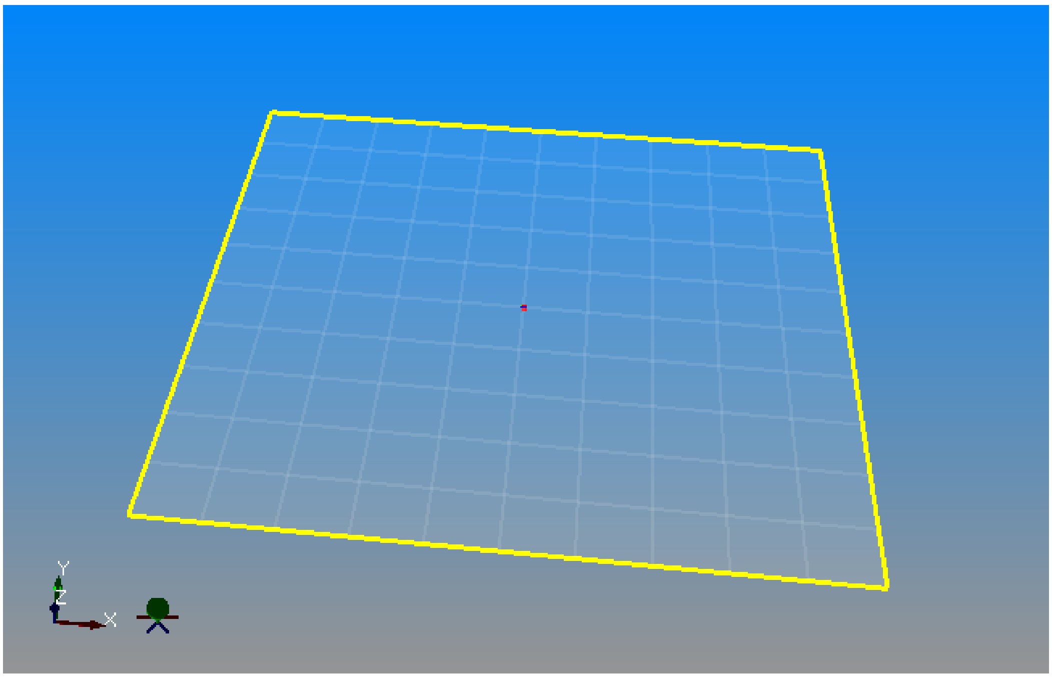
To **delete** a Group you have two ways:

- open the contextual menu of the group (click on RMB) and select *Remove Group* command
- starting from a selection of (also) several objects click *[Canc]* on keyboard. According to the selection, a different

When deleting a not empty group with Canc, it is asked if you want to delete the objects inside or not.

⚠ Don't remove a not empty group if included in a heterogeneous multi-selection.

Planes



Reconstructor® allows you to define planes in your project.
A plane has a width and a height that can be specified in the property editor window under *Extent on X* and *Extent on Y*. A plane has also all the properties of a project entity:*Color ID, comment, hyperlink, Draw annotation, Draw coordinate frame*. To enable the last one is particularly useful in order to visualize the plane's normal direction.

From the contextual menu of a plane item, you can also start the following commands:

- *Cross section*: use this plane to compute a cross section of the models activated in the scene. The result is a polyline
- *Invert direction*: inverts the plane's coordinate frame Z axis direction. The positive direction determined by the plane has an influence for instance in cross sections, volume and cut and fill calculations.
- Meshing->*Create mesh from a plane: to create a mesh starting from the plane object. Useful for example for inspection tool.*
- *Set costrained draw*: to pick points in the 3D scene and draw them as a projection on the plane.

See also Project items and Plane creator/editor.

Point Clouds

There are two different kinds of point clouds in Reconstructor®:

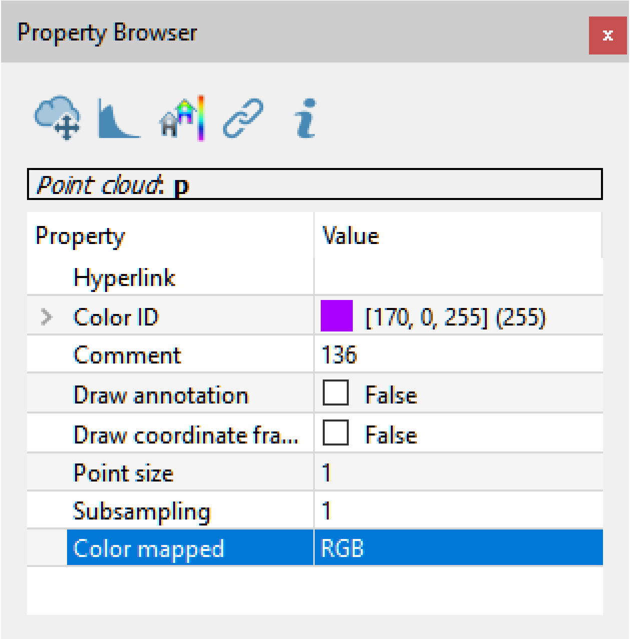
- ***Structured*** point clouds (also called *grid point clouds,* or *range images*, or *scans*): scan-based clouds, "fixed-origin" or tripod scans. They include row/column info and the origin scan's position.
- ***Unstructured*** point clouds

Operations available for both unstructured and grid point clouds are:

- **Recompute bounding box**: useful sometimes when the bounding box is not updated (available only if the model is loaded in memory)
- **Histogram**: the histogram tool allows to optimize the contrast by histogram stretching (available only if the model is loaded in memory and if the current color type is 1f or 1d, i.e. a high-dynamic single-channel point color)
- **Colors mapping**: point colors can be remapped to a pseudo colored scale to improve the dynamic range, if the histogram stretching is not sufficient (available only if the model is loaded in memory and if the current color type is 1f or 1d, i.e. a high-dynamic single-channel point color). This tool can be also used for range segmentation and inspection.
- **Inclination from plane**: if the Inclination color type is available (the model must be pre-processed) the dialog allows to select a plane whose normal is projected to the normals of the model (dot product). The result is added as a 1f color type.

Operations available only for grid point clouds are:

- **Edit 2D**: edit the grid
- **Extract edges**: create a polyline from the computed orientation and depth discontinuity during the pre-processing phase. The polylines are as contiguous as possible, adjust the max search distance and minimum edge length to optimize the outcome.
- **Cross Sections**
- **Multiresolution Mesh**
- **Simplify Points**: to determine the most relevant points from a point of view of shape description, saving them into a new unstructured point cloud.



The editable properties - for both of them - are:

- *Color ID*: you can edit the color ID by clicking on the *value* field of the color ID property. To render the items with their color IDs, select *Navigation->Color by ID.*
- *Draw coordinate frame*: if true, draw the local coordinate frame
- *Point size*: size of drawn points
- *Subsampling*: subsampling rate of rendering of the cloud of points
- *Color mapped*: select the color type to render the point cloud

Advanced notes

A Grid point cloud of point can be viewed as “image” where the number of pixels (pix) are equal to the number of point the scanner has acquired (valid and invalid).

1	452	-0.026171	-3.7225	-7.1416	104	104
1	455	-0.026171	-3.7225	-7.1416	171	171
1	459	0.16056	-3.0731	-1.6781	70	70
1	460	0.16513	-3.0732	-1.6781	129	129
1	461	0.16969	-3.0733	-1.6782	174	174
1	462	0.17425	-3.0734	-1.6786	174	174
1	463	0.17882	-3.0735	-1.6787	168	168
1	464	0.18337	-3.0737	-1.6792	171	171
1	465	0.18793	-3.0738	-1.6795	168	168
1	466	0.19246	-3.074	-1.6803	154	154

2D pixels of the Grid

3D points

RGB val

Point clouds data storage

Data is stored in an XML description file + pure binary vector files. The file extension of the binary data specifies the format, where data.NT: N=num of components (1,2,3,4...), T=data type (byte, short, word, half, float, double).

For point colors, a special 3bc (c stands for compressed dynamic range) file is created for rendering only, for instance after histogram equalization.

For example, the unstructured point clouds *myUnstruct* will be saved in the following files in the project folder:

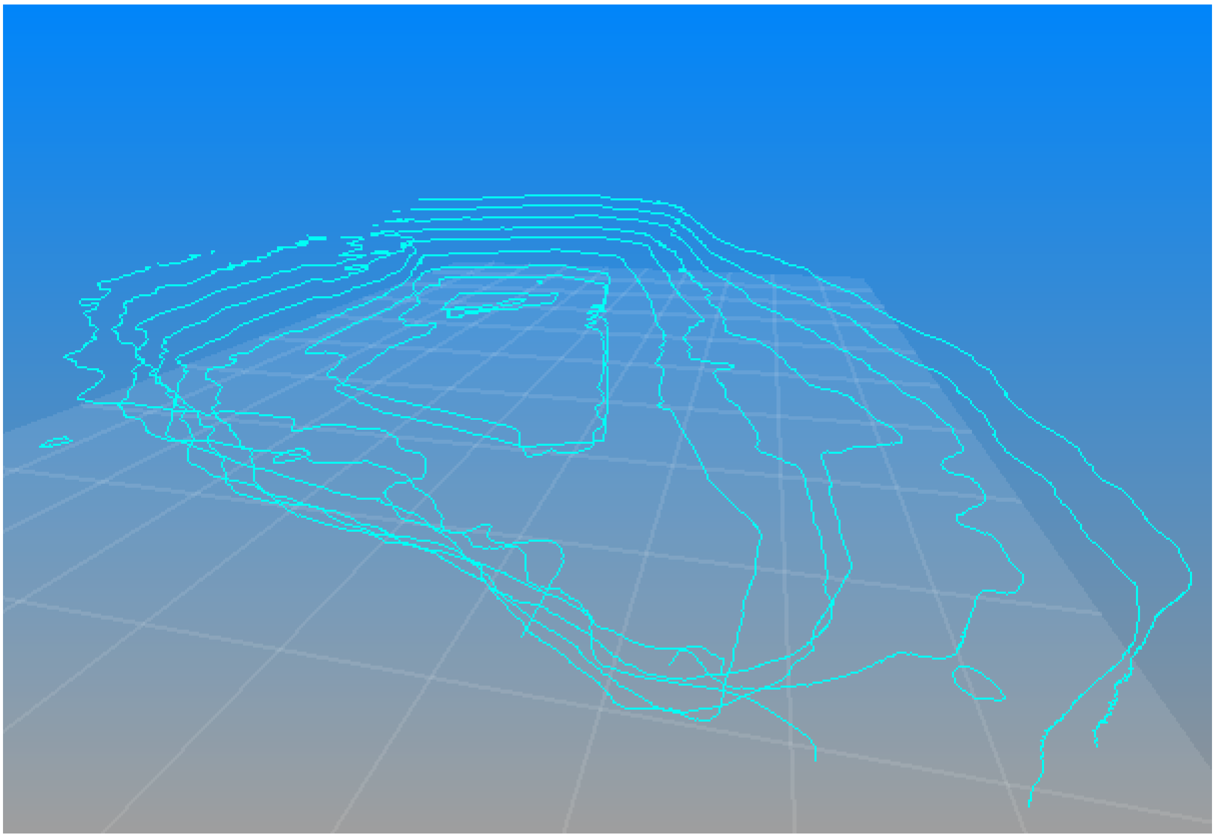
- Unstructs/myUnstruct.rup, an XML file containing the cloud's properties
- Unstructs/myUnstruct_rup/points.3f
- Unstructs/myUnstruct_rup/status.1i

- Unstructs/myUnstruct_rup/Reflectance.1f
- Unstructs/myUnstruct_rup/Reflectance.3bc
- ...

The grid point cloud *myGrid* will be saved in a similar way in the folder *Grids* inside the project folder. Grid point cloud files have extension ".rgp"

An unstructured point cloud, instead, is like an unorganized list of points.

Polylines



Polylines are sets of open or closed polygons that appear in your 3D scene. You can create polylines in several ways:

- From the points you have collected in the point list window.
- From the cross sections tool (the polylines created by this tool are saved in your project in the *Cross sections* group).
- From the command *Data management → Meshing → Get mesh borders as polylines*, belonging to the context menu of the triangle meshes.
- From the Mesh Crest and Toe tool.
- From the edge extractor of grid point clouds, available via *Data management → Cloud processing → Extract edges*
-

The most important context command for polylines is the *Export* command, that allows you to export the polyline in the DXF format, readable by AutoCAD®.
It is often useful to define polylines in order to delimit the region in which you want to

calculate volume, or cut and fill volume. Polylines can be also used to select points or to cut meshes.

Projectors

A projector is a camera view plus an image to project on the scene. It can colorize whatever geometry is “lighted” by the projector allowing also to see occlusions if shadow mapping is supported.

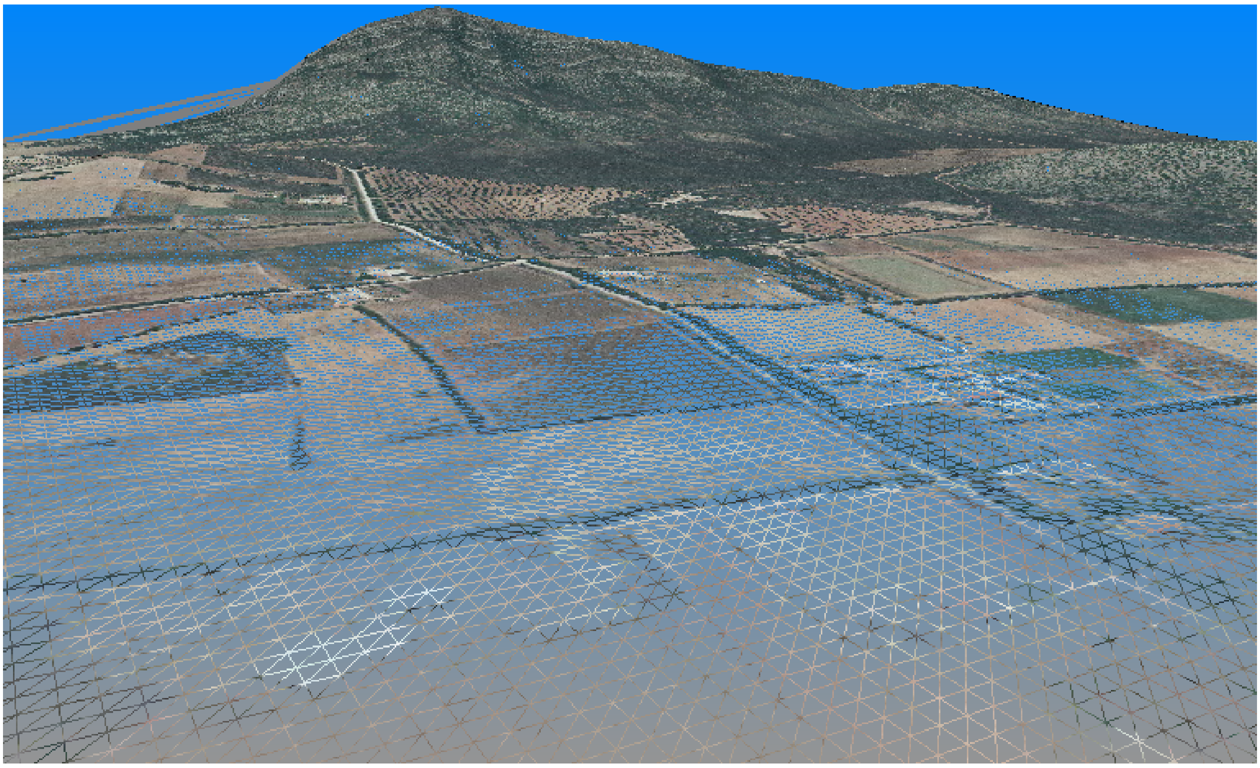
The specific commands are:
Load projector: to load (light on) the image on the 3D model.
Set projector image: to set the desired image to project
Edit projector image: edit the projector’s image to set the transparent pixels in order to produce better blending among the projectors

The specific editable properties are:
Activate: the image is loaded and projected on the scene

Shadow mapping

This technique works only for triangle meshes. To compute the occlusions from the view point of the projector, a depth buffer of the scene is created. Unfortunately the accuracy of the shadows depends on the sampling of this depth buffer, thus its width and height and depth range. Ideally the size should be infinite, but this cannot fit in the memory so the resolution must be tuned. It's recommended to use at least the width and height of the projected image and to keep the clip near and far planes as close as possible to the desired scene depth range. Because of this coarse resolution, the resulting depth buffer is affected by quantization noise. To compensate for this, an epsilon term is used as a tolerance for the depth comparisons. For each depth point, the epsilon formula (known as polygon offset) is: ShadowScale•Depth + ShadowBias. If some random black points are visible when casting the shadows, try to increase the scale and bias, but pay attention that excessive values produce less shadows.
Shadow bias: additive term for polygon offset during the depth buffer generation (default is 16)
Shadow scale: multiplicative factor for polygon offset during the depth buffer generation (default is 2.5)
Shadow width/height: size of depth buffer used to compute the occlusions from the view point of the projector. It's recommended to use at least the same size of the projected image, displayed in the tooltip of the projector.

Triangle Meshes



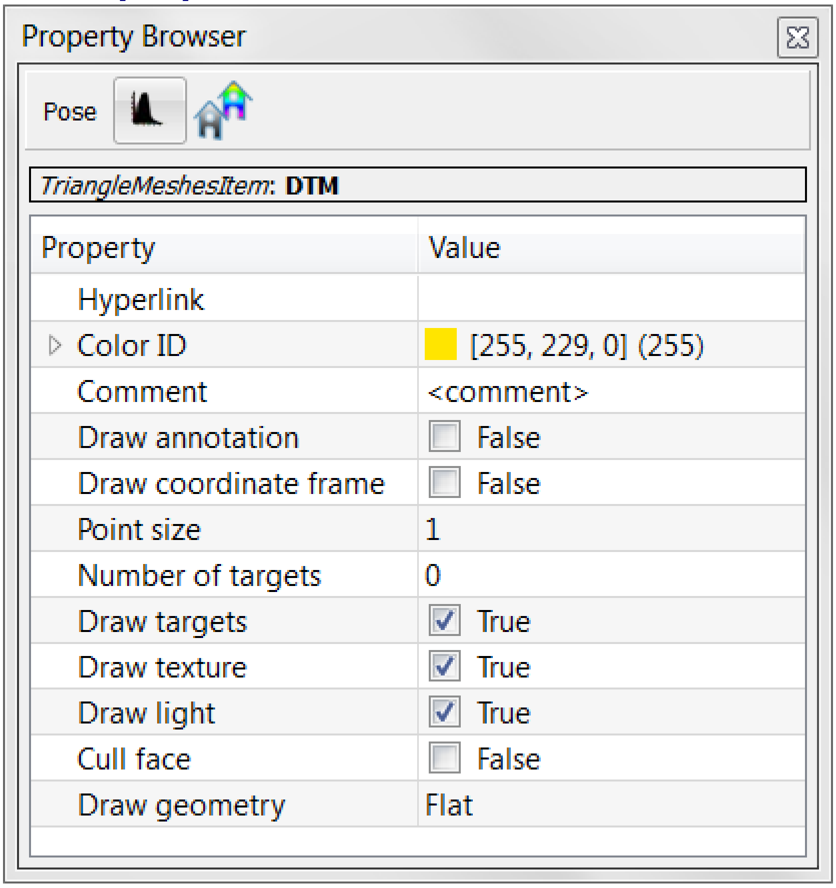
A triangle mesh is a 3D model represented by a set of triangles connected by common edges and common vertices. If a point cloud includes a set of points, a mesh includes more information because it stores how the points are connected.
A triangle mesh therefore defines a surface in the 3D space. The mesh's edges that are owned by only one triangle are called mesh borders. A mesh is called closed if there are no borders. A mesh is called manifold if all its edges are shared by at most two triangles, and a mesh is called non-manifold if there are edges that belong to three or more triangles. A mesh is called watertight if it has no holes, or if its borders are only one closed outer polygon.
To each vertex of the mesh, a color can be associated. Each mesh's triangle is rendered with a gradient of the three colors of its vertices. The rendering color may depend also on the triangle's inclination with respect to the light source.
Reconstructor® has various techniques for generating and editing meshes. Each triangle mesh item can contain more than one submesh.

Actions available from meshes' context menu

- **Meshing->Compute normals**: computes or updates the triangles' normals for the mesh.
- **Meshing->Invert winding**: inverts the ordering of the vertexes for each triangle, so the surface is flipped to the opposite side and also the normals are inverted.
- **Meshing->Compute area**: r eturns the mesh area as sum of the areas of all the mesh's triangles.
- **Meshing->Compute volume from Z=0 plane**: returns the volume resulting from integrating the mesh on the XY plane of the current UCS. Mesh triangles below the XY plane will result in zero volume.
- **Meshing->Get mesh borders as polyline**: creates a new polyline containing the mesh's borders, and adds it to the project.
- **Meshing->Make single mesh**: to cluster in a single mesh an arbitrary set of triangle meshes.
- **Meshing->Convert to point cloud**: create an unstructured point cloud from the vertexes of the mesh using the color attribute of the mesh.

- **Meshing->Mesh editor**: opens the mesh editor dialog.
- **Photo&Color->Sample texture at vertexes**: sample the color of texture (if available) at the vertexes of the mesh and store as the color attribute of the mesh.
- **Photo&Color->Create texture map**: compute a blended texture map for the mesh from the projectors.

Meshes' editable properties



- *Color ID:* False color that you can activate with *Screen settings -> Color by ID* command
- *Draw coordinate frame*: if true, draw the local coordinate frame
- *Point size*: if draw geometry is set to points, they are draw with this size
- *Draw texture*: if available, the texture is loaded and rendered
- *Draw light*: if the normals are valid, the triangles are rendered with smooth shading
- *Cull face*: if true, the triangles are rendered only if the normal faces the view direction, and backface triangles are hidden. If False both sides of the mesh are visible
- *Draw geometry*: select flat, wireframe (only edges), or only vertexes (points) to display them

Advanced Options

Triangle meshes data storage

Data is stored in an XML description file + pure binary vector files subdivided in folders for each submesh. The file extension of the binary data specifies the format, where data.NT: N=num of components (1,2,3,4...), T=data type (byte, short, word, half, float, double). If a mesh is called myMesh, a typical way of storing it will use the following files in the project folder:

- Meshes/myMesh.rtm
- Meshes/myMesh_rtm/submesh/vertexes.3f

- Meshes/myMesh_rtm/submesh/triangles.3i
- Meshes/myMesh_rtm/submesh/...
-

See also Meshing techniques and Mesh editing.

User Coordinate System (UCS)

User coordinate systems are a central concept in Reconstructor®. In the same project, many coordinate systems can exist. An empty project contains at least a default UCS, called Main.

To create a new UCS, you can click on any project item and select *Create UCS from this pose*. The new UCS is added to the User Coordinate Systems group in the project window.

The current UCS

The Figures 1, 2 and 3 below show the same scene with three different UCS's set as *current*. The current UCS is visible from the long coordinate axes (the red is the X axis, green is the Y, blue is the Z axis).

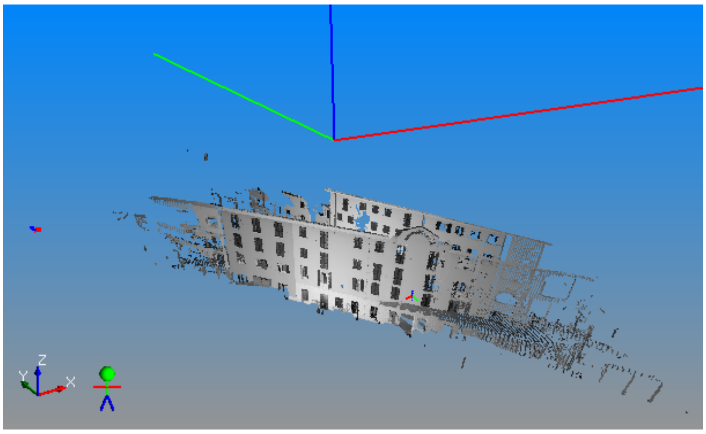


Figure 1

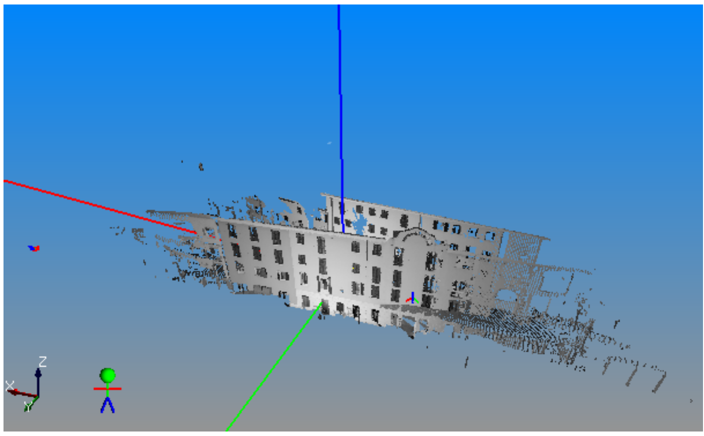


Figure 2

There is always one and only one current UCS. This is the UCS displayed in bold in the User Coordinate Systems group in the project window. To change the current UCS, right-click on another UCS and select set as current.
The current UCS is the current coordinate system for all the operations in Reconstructor®.

- Point coordinates in the Readout Window are displayed in the current UCS.
- Model position in the Adjust Pose window is displayed in the current UCS.
- Points in the Point List Window are displayed, imported and exported in the current UCS.
- All project data (point clouds, meshes, polylines, etc.) are imported interpreting their position in the current UCS, and exported saving their position as in the current UCS.
- The pose matrix shown in the Pose Dialog is referred to the current UCS.
- The navigation system will perform "human" rotation movements assuming that the horizontal plane is the XY plane of the current UCS (see the option *Enable human movements while rotating* in Navigation Options for more information).
- Many Reconstructor® functions require a vertical direction and altitudes to be defined in your 3D scene. *Cut and fill calculation* is an example of these functions. The general assumption in Reconstructor® is that the current UCS's Z axis defines the vertical direction, and the altitudes are the distances of project items from the Z=0 plane of the current UCS.
- The *Create/Edit Plane* command allows to make a plane horizontal or vertical. Horizontal and

Vertical directions, again, are always defined by the current UCS.

- Etc.

Therefore, if the user changes the current UCS, all the above-mentioned windows and functions will be affected in the way they visualize the coordinates or compute the models' positions.

UCS can be also created from the registration between two point lists.

You can change UCS dimension, open *Display and navigation options* -> *Object reference frame size [m]* and insert the desired size.