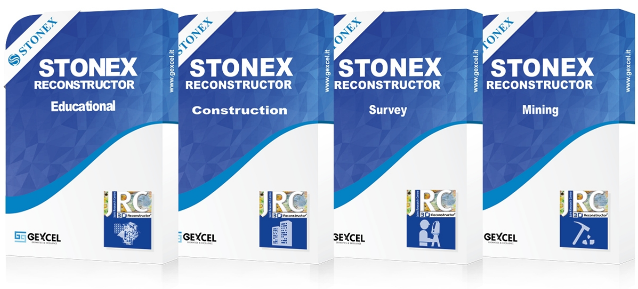
STONEX-RECONSTRUCTOR®

"Power and flexibility with your STONEX data!"

SOFTWARE MANUAL



GEXCEL GEOMATICS & EXCELLENCE

Gexcel Srl
Central Admin. | Sales | R&D Lab
c/o CSMT, Via Branze 45
I-25123 Brescia, Italy

Registered Office | R&D Lab
c/o Polaris Science and Technology Park
Loc. Piscinamanna
I-09010, Pula (CA), Italy

Table of contents

STONEX Reconstructor Manual

STONEX and Gexcel have developed and optimized the JRC 3D Reconstructor® software to perfectly support the survey projects carried on with the Stonex X300 laser scanner.

JRC 3D 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. JRC 3D Reconstructor® imports all the most used laser scanner formats in the market. The workflow begins with the **LineUp® Tool**, an automatic and very robust 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, JRC 3D 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, etc.

Please click on one of the following sub-topics:

- Contents
- System Requirements
- Installation
- License Manager
- Disclaimer
- *End User License Agreement*

Contents

The present online 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 JRC 3D Reconstructor®

iii. The main features of the user interface

iv. An explanation of the functionalities of JRC 3D Reconstructor®, according to the voices in the menu. These are therefore the main chapters of this section:

- File
- Navigation
- Outputs
- Tools
- Windows
- Workspaces
- Help

Note: some commands are not available in all the workspaces (pay attention to your license)!

System requirements

System requirements - minimal

- OS: Windows (XP SP2, Vista, 7, 8, 10)
- Version: 32 bit
- Graphic card: NVIDIA GeForce with 512MB at least
- RAM: 4GB at least

System requirements - recommended

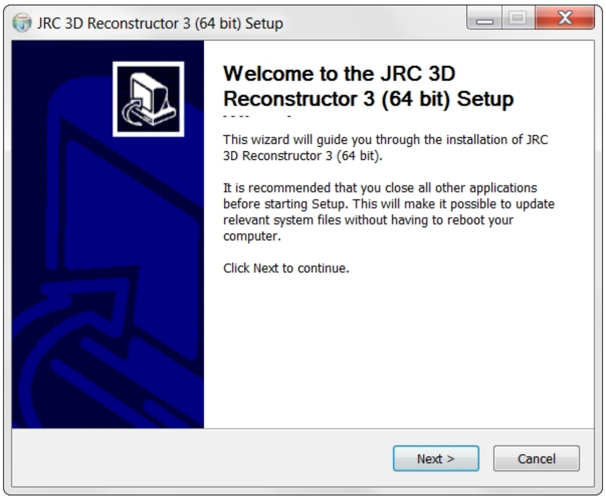
- OS: Windows (XP SP2, Vista, 7, 8, 10)
- Version: 64 bit
- CPU: multi-core processor (8 Cores at least)
- Graphic card:
    - NVIDIA GeForce GTX with 2GB Ram (for a large use of points)
    - NVIDIA Quadro (for a large use of mesh and texture)
- RAM: 16GB

JRC 3D Reconstructor® compiled for 32 bit processors can only address 4GB of RAM memory. For medium/large size projects, we strongly recommend the 64 bit version. The graphics card must be at least OpenGL 1.3 compliant with ARB_transpose_matrix, ARB_multitexture, ARB_pbuffer extensions. Other extensions are automatically used if hardware support is found, resulting in better performance and increased functionality.

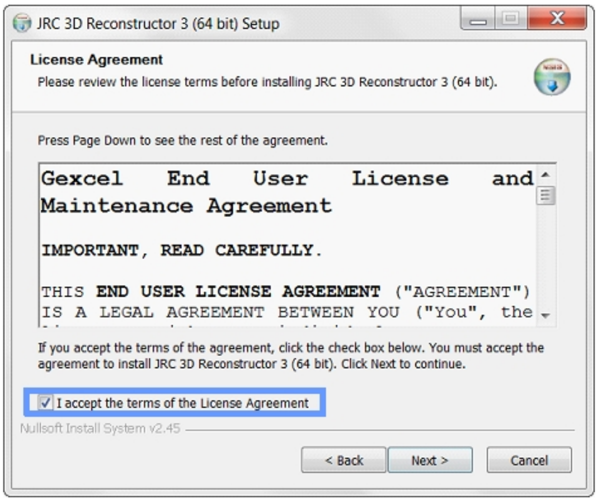
Installation

It's highly recommended both to deactivate all the Antivirus and Antispyware software and to keep closed any JRC 3D Reconstructor® application **before starting the installation**.

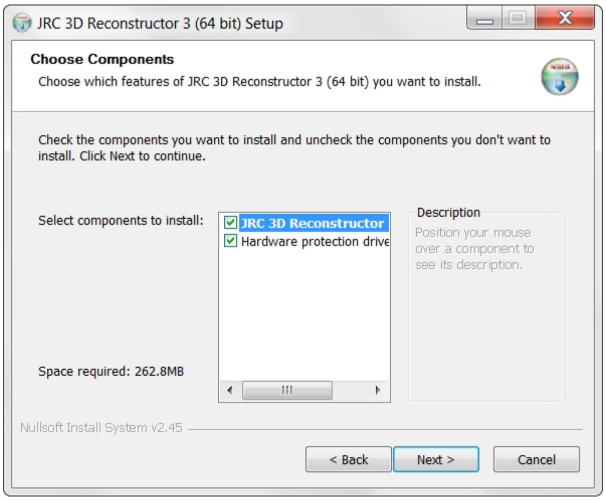
1. Run *Reconstructor_3.2.1.xxx_win64.exe* (the installation file, only given as an example) and follow the steps.



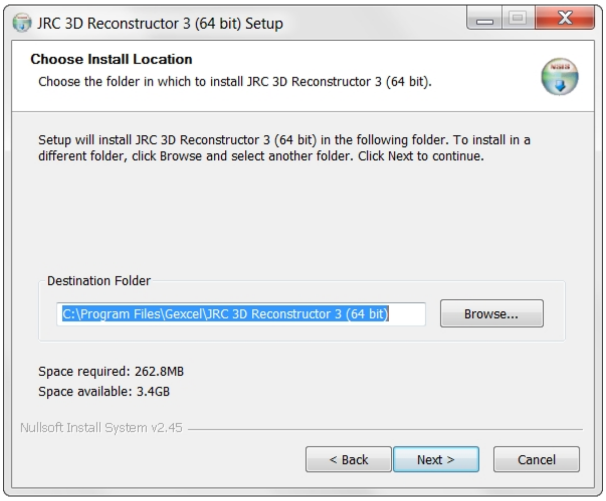
2. Accept the **License Agreement terms** and follow the steps.



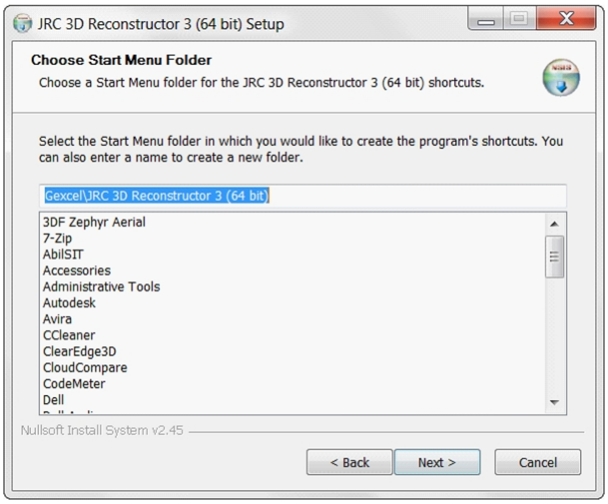
3. Select the **components** you want to install. Hardware drivers must be always installed!



4. Choose Install Location (Default path is *C:\Program Files\Gexcel\JRC 3D Reconstructor 3 (64 bit)*)



5. Choose Start Menu Folder and follow the steps.



Installation starting.....

See also *Help* → *About JRC 3D Reconstructor®* to find all the information about the license, expiration date included.

Multi-licensing (USB licenses)

To activate the multi-seat licenses, you must install the software on all the computers to be used. The computers can be positioned in different locations, but they must be connected to the same network (same IP address and subnet mask).
Insert the USB key into one of the computers. Once opened JRC 3D Reconstructor® on a computer, the license will be deducted from the multi-seat license.

License Manager

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

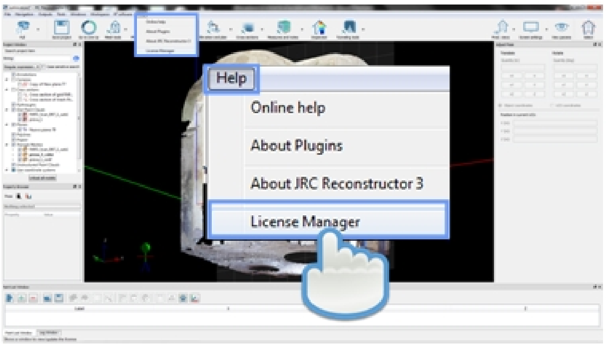
- SOFTWARE LICENSE (SL) - FIRST ACTIVATION
- SOFTWARE LICENSE (SL) - UPDATING
- USB LICENSE - ACTIVATION/UPDATING

SOFTWARE LICENSE (SL) - FIRST ACTIVATION

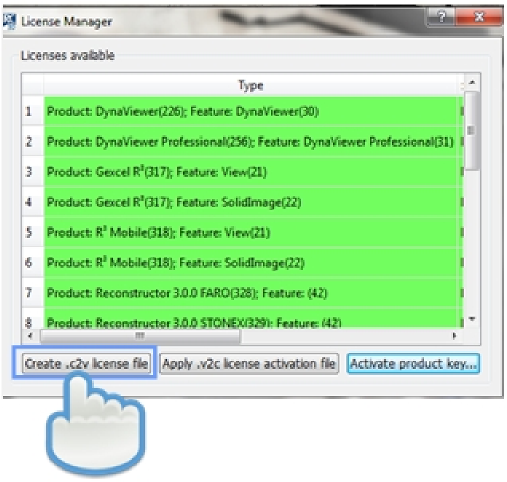
Here it's possible to learn how to activate the software based license SL (on single computer) by

1. Generating a **.c2v** file (necessary to get the .v2c file from Gexcel)
2. Applying a **.v2c** file (necessary to activate your software license)

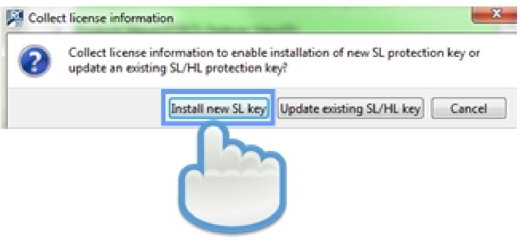
1. Open the software and select *Help* → *License Manager* in order to create a .c2v file.



Click on *Create .c2v license file*.



Click on *Install new SL key*.

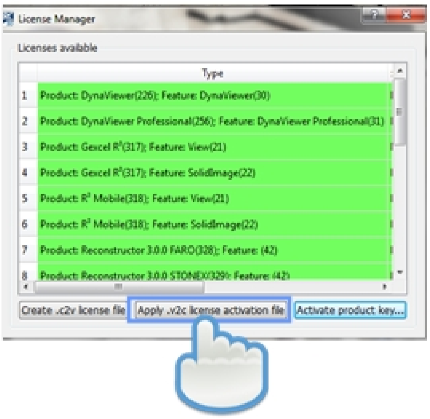


Type the desired file name and *Save* the Fingerprint file (.c2v).

Send the .c2v file by e-mail to license@gexcel.it. It's highly suggested to send this file in a zipped format.

1. Open the software and select *Help* → *License Manager* in order to apply a .v2c file.

   Click on *Apply .v2c license activation*.



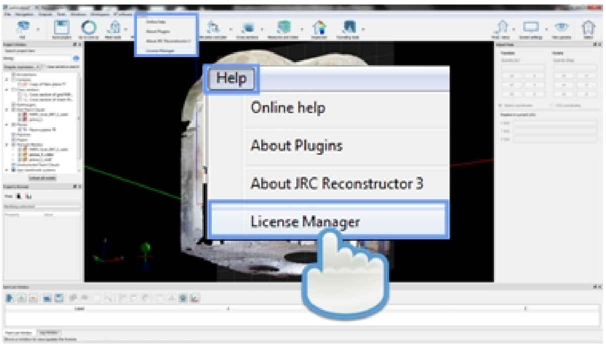
Select the .v2c file (received from license@gexcel.it) and *Open* it.

SOFTWARE LICENSE (SL) - UPDATING

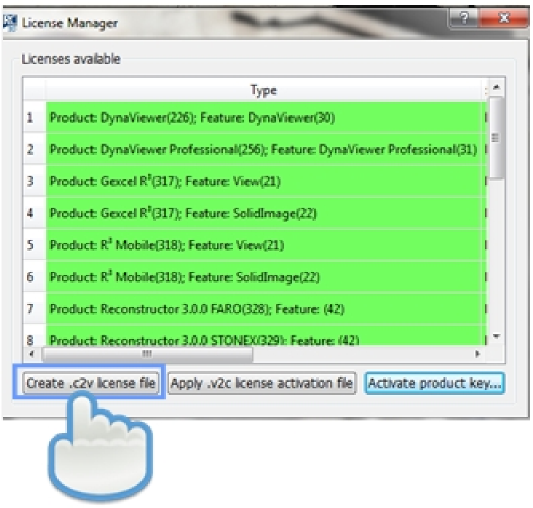
Here it's possible to learn how to update the software based license SL (on single computer) by

1. Generating a **.c2v** file (necessary to get the .v2c file from Gexcel)
2. Applying a **.v2c** file (necessary to update your software license)

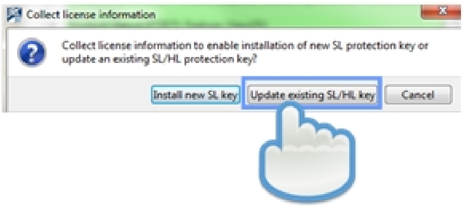
1. Open the software and select *Help* → *License Manager* in order to create a .c2v file.



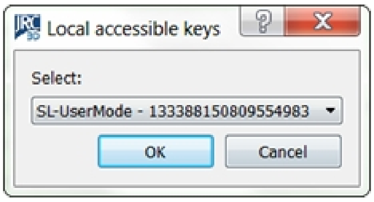
Click on Create .c2v license file.



Click on *Update existing SL/HL key*.



Open the drop-down menu to look at the *accessible keys* (activations) available on your computer.



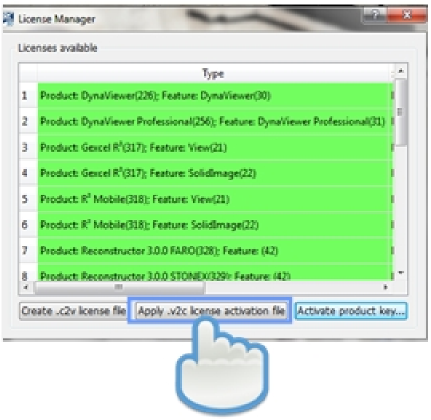
Select the SL to update.

Type the desired file name and *Save* the Fingerprint file (.c2v).

Send the .c2v file by e-mail to license@gexcel.it. It's highly suggested to send this file in a zipped format.

2. Open the software and select *Help* → *License Manager* in order to apply a .v2c file.

   Click on *Apply .v2c license activation*.



Select the .v2c file (received from license@gexcel.it) and *Open* it.

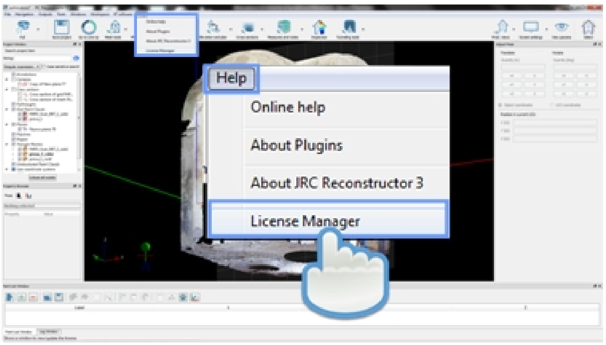
USB LICENSE - ACTIVATION/UPDATING

Here it's possible to learn how to activate/update an USB dongle (!) HL *(with USB plugged in the computer)* by

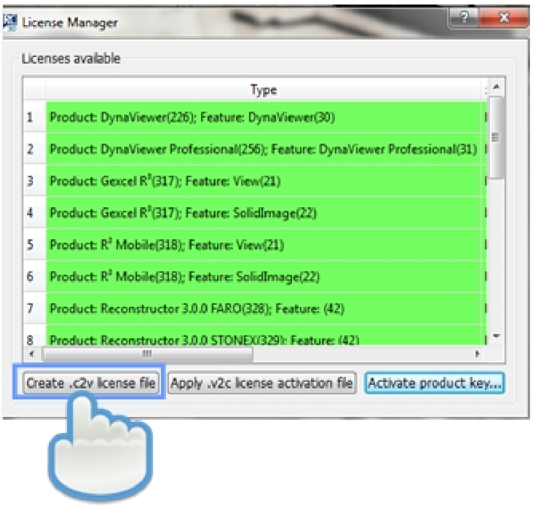
1. Generating a **.c2v** file (necessary to get the .v2c file from Gexcel)
2. Applying a .**v2c** file (necessary to activate your USB license)

(!) Pay attention: for USB dongles, the first activation and the updating follow the same procedure.

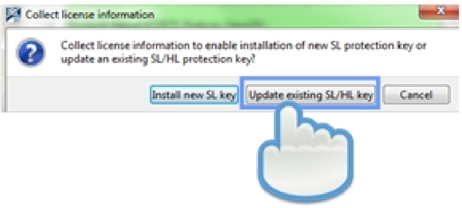
1. Open the software and select *Help* → *License Manager* in order to create a .c2v file.



Click on Create .c2v license file.

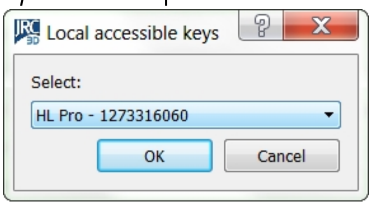


Click on *Update existing SL/HL key*(!).



(!) Click here also for the first activation

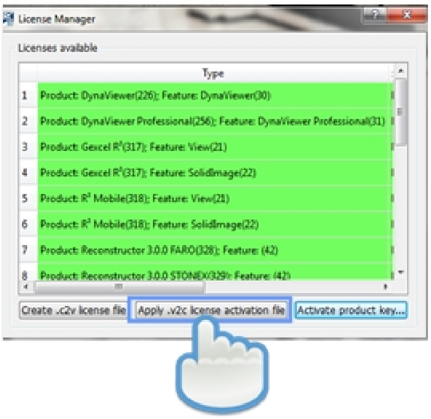
Open the drop-down menu to look at the accessible keys (activations) available on your computer.



Select the HL (hardware license): it represents the USB dongle [*remember to plug in the USB dongle before starting the software*].
Type the desired file name and *Save* the Fingerprint file (.c2v).
Send the .c2v file by e-mail to license@gexcel.it. It's highly suggested to send this file in a zipped format.

2. Open the software and select *Help* → *License Manager* in order to apply a .v2c file.

   Click on *Apply .v2c license activation*.



Select the .v2c file (received from license@gexcel.it) and *Open* it.

At the end of the license activation process, reboot JRC 3D Reconstructor®.

Software Language

JRC 3D Reconstructor® supports the following languages: English (default), Italian, Chinese, Japanese, Spanish.

JRC 3D Reconstructor® automatically detects which language to use by looking at your operating system's properties. If your language is not available, English setting is activated automatically. To change language just modify the language setting of your computer.

Disclaimer

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.

See also End User License Agreement.

End User License 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 GEXCEL SRL ("Gexcel") 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"). This Agreement will also apply to any Software error corrections, updates and upgrades subsequently furnished by Gexcel, 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.

Gexcel may change the terms of this Agreement from time to time (and shall upload such updated terms 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, Gexcel 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 Gexcel 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 Gexcel's consent, You hereby agree that, before exercising any such rights, You shall first request such information from Gexcel in writing detailing the

purpose for which You need the information. Only if and after Gexcel, at its 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 Gexcel 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 Gexcel, or issues to Gexcel 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 Gexcel of the applicable Fees for the relevant license, Gexcel 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 Gexcel, as that term is defined herein. Gexcel 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 th evaluation time please contact sales@gexcel.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 Gexcel 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 (with an exception for any limited money-back guarantees that are specified in writing or on the Gexcel website, www.gexcel.it, at the time of purchase of a license.) 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 Gexcel's net income. When Gexcel has the legal obligation to pay or collect such taxes, the appropriate amount shall be invoiced to and paid by You unless You provide Gexcel 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, Gexcel shall provide Software Maintenance Services, including both technical support services and software releases. (1) Technical Support Service is provided through e-mail Address: support@gexcel.it. (2) As part of its maintenance services, Gexcel 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 Gexcel's web site (www.gexcel.it) or delivered on digital support upon written request to Gexcel.

4.2. Software Maintenance Limitations. (1) Gexcel shall have no obligation to provide Releases or Support for the Software except as set forth hereafter. Gexcel shall not have any responsibility to develop subsequent components for the Software or additional processes for You, except as explicitly set forth herein. (2) Gexcel shall have no obligation to fix errors in the Software within a specific time duration. Gexcel 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 Gexcel) 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 Gexcel, 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 Gexcel. In the event that Gexcel 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) Gexcel 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 Gexcel 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, Gexcel 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 Gexcel 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 to Gexcel the damaged dongle and pay to Gexcel a Replacement Fee of 250,00 Euro (delivery charges excluded), as advised by Gexcel. In case of USB Dongle Key loss, or stolen, You shall provide Gexcel with a written and signed declaration, contact Gexcel at sales@gexcel.it and pay to Gexcel 80% of the software price for a new Dongle Key.(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 Gexcel with a written and signed declaration, contact Gexcel at sales@gexcel.it and pay to Gexcel 80% of the software price for a new Dongle Key.

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 Gexcel.

5. NO WARRANTIES.
THE SOFTWARE IS BEING DELIVERED TO YOU "AS IS" AND GEXCEL MAKES NO WARRANTY AS TO ITS USE OR PERFORMANCE. GEXCEL 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, GEXCEL 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.
GEXCEL 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 GEXCEL 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 GEXCEL OR AN AUTHORISED GEXCEL'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 GEXCEL'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 GEXCEL 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.
GEXCEL 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 Gexcel: (a) any information, in whatever form, disclosed by Gexcel 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 Gexcel 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, Gexcel may terminate this Agreement immediately and without notice if You breach any provision of this Agreement or of any other agreement between You and Gexcel. Upon termination or expiration of this Agreement: (a) any fees owed to Gexcel 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 Gexcel, 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.

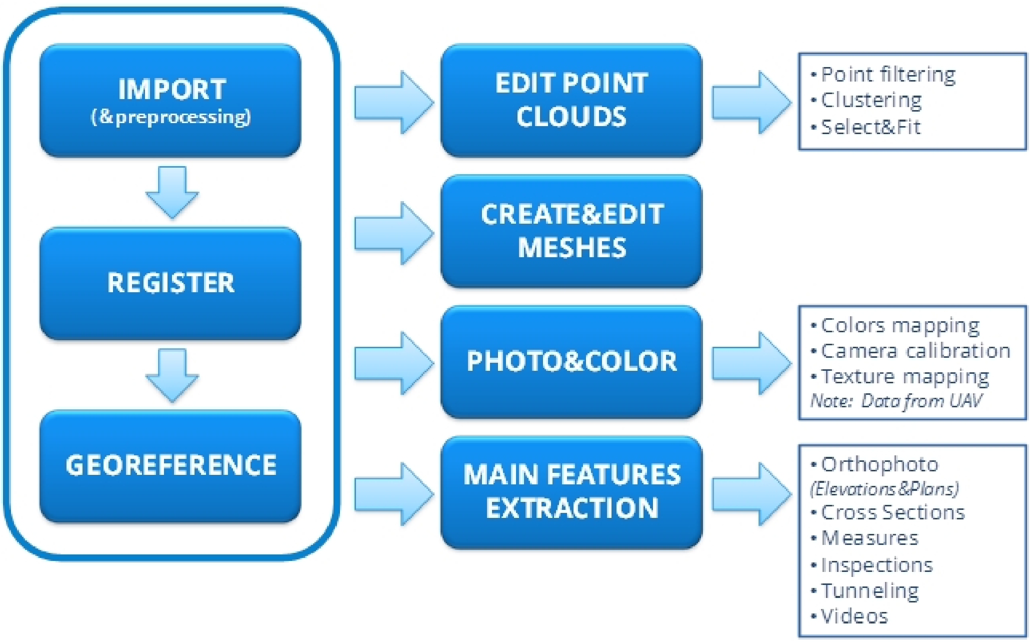
9. EXPORT RESTRICTIONS.
You shall not export, directly or indirectly, the Software (or any component part or output thereof) to any country in violation of any law or regulation, including without limitation any law or regulation of the U.S. Government or any agency. You are solely responsible for complying 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 Gexcel 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 Gexcel may use Your company name on Gexcel's customer list provided to third parties.

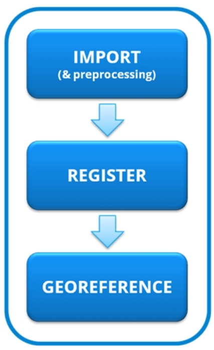
11. GENERAL.
This Agreement shall be governed by the laws of Italy without regard to conflicts of law principles. The Courts of Italy will have exclusive jurisdiction over any disputes arising under this Agreement and the parties hereby irrevocably consent to the personal jurisdiction of such courts. 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 Gexcel 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 Gexcel (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 JRC 3D Reconstructor®.
First you need to create a New project or Open an existing project.
Each project has its own folder called with the same name. JRC 3D Reconstructor® projects contains at least some of the following subfolders: *Imports/ Exports/ Unstructs/ Grids/ Meshes/ Polylines/ Trash/ Images/ Movies/*. These folders are automatically created when the project gets populated.
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 in LineUp® environment.

What's LineUp®?

LineUp® is a tool in JRC 3D Reconstructor® that easily allows you to:

- Import
- Preprocess
- Register
- Georeference

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

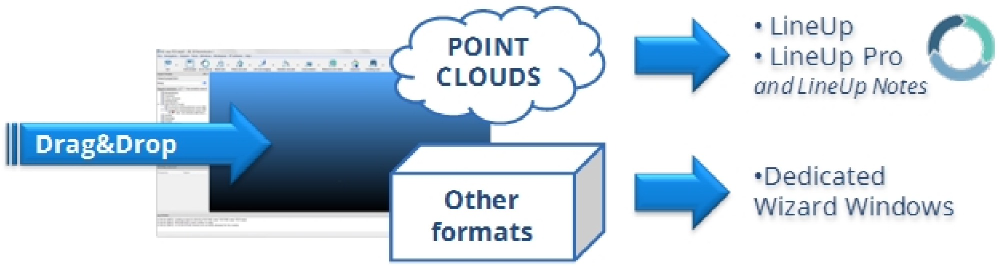
Note that you can also register and georeference not only point clouds but also 3D mesh models.

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. Main features extraction (analysis tools)

Import

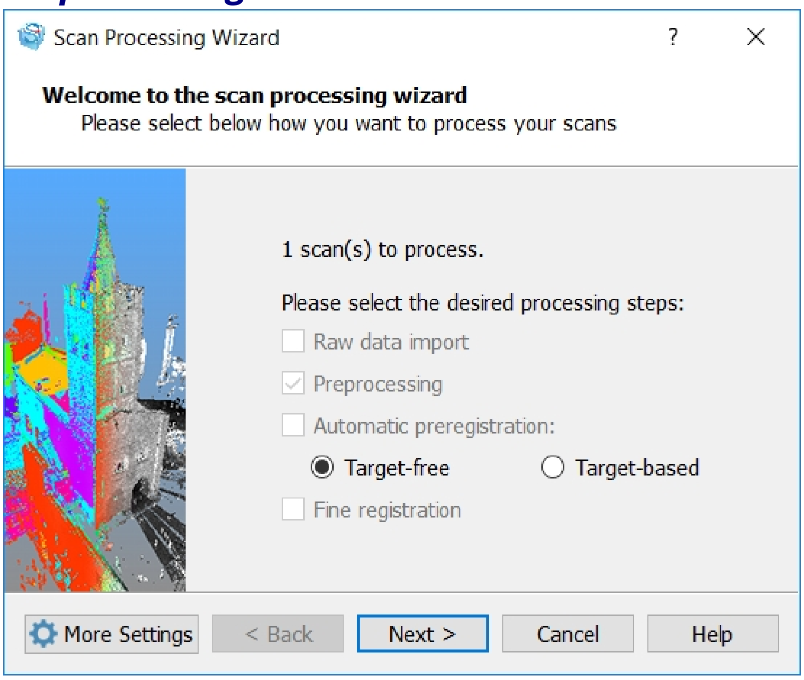
To quickly import data, just drag and drop the files you want to import anywhere in the 3D scene. A wizard will appear to ask you for import options, depending on kink of data.



a. If you're importing a point cloud the **LineUp® Tool** 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.

a. Importing Point Clouds...& Preprocessing

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

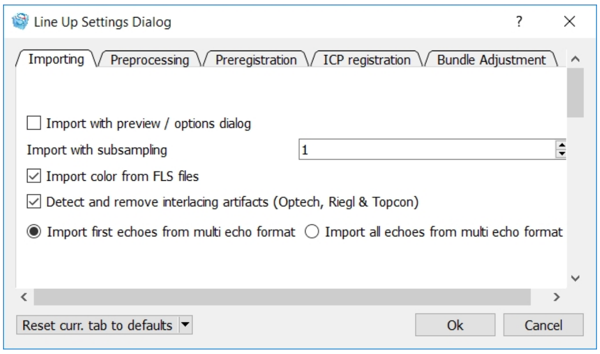


You can follow its instructions step by step, in order to import raw data and to preprocess the scans.

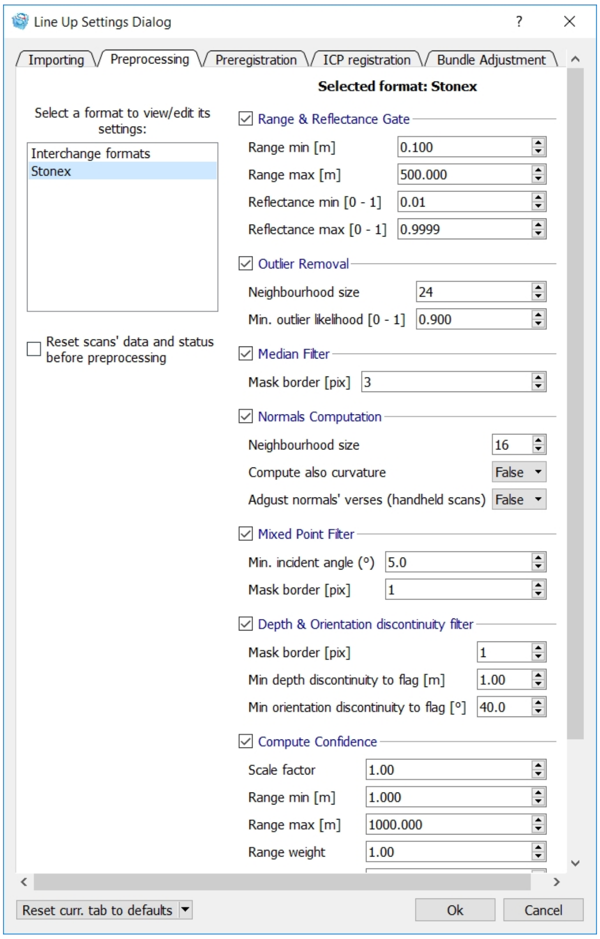
Note: it's also possible to register (both preregistration and fine registration) the imported point clouds using the same wizard.

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.
- The Stonex Data Importer window help you to colorize your scans during importation.

b. Importing "Other Formats"

- If you're opening one or more of these objects:
  - Triangle meshes
  - Polyline/trajectory
  - CAD Models

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

- If you're opening a

Reconstructor project

all the scans inside it (and other items, as in JRC 3D Reconstructor® projects) will be loaded in your current project.

At the end of both of these import phases, your models will be loaded in your project and saved in JRC 3D Reconstructor® internal format.

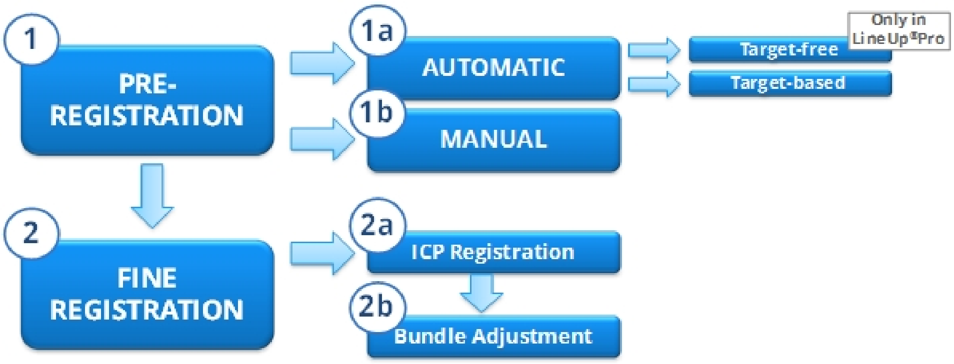
See details in Import section.

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

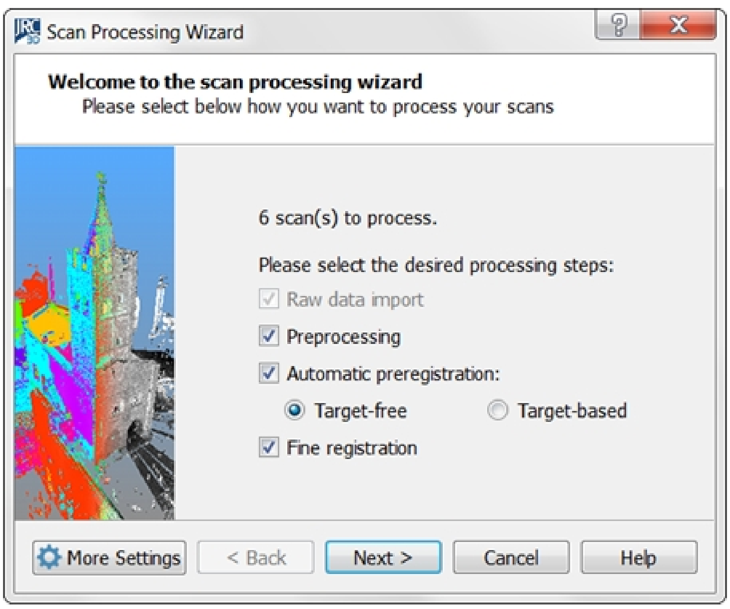
Register

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.



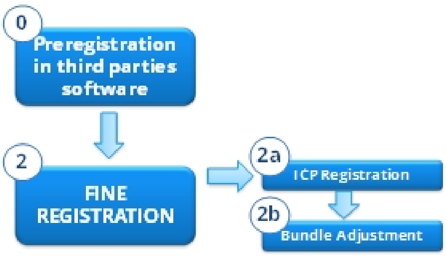
Start LEVEL You can simply use the *Scan Processing Wizard* (also simultaneously with the import phase) and check all the steps, using default parameters.

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.

WF 1

After importation of a pre-registered project



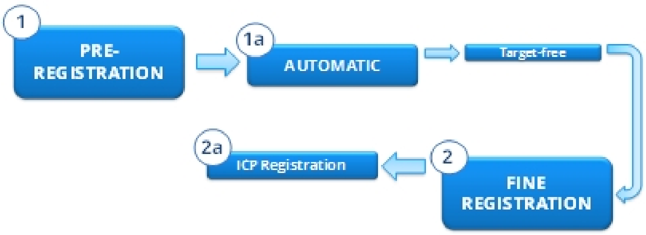
If you pre-registered your scans with a third part software (as the owner laser scan) you can import the just aligned scans and simply make a fine registration, in order to further reduce the registration error.

WF 2

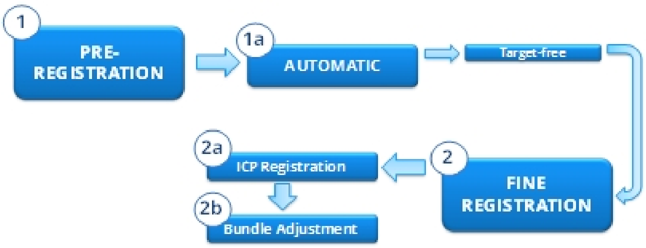
Automatic registration without targets (*only with LineUp® Pro*)

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



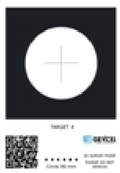
or also a bundle adjustment



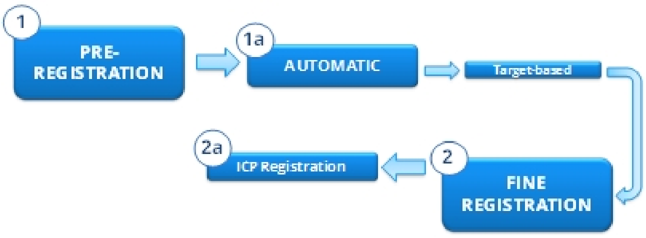
WF 3

Automatic registration using targets

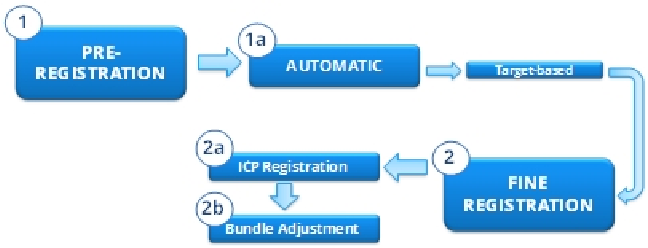
If you made a survey using Gexcel's targets, a special function automatically register a set of scans by detecting and matching targets.



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



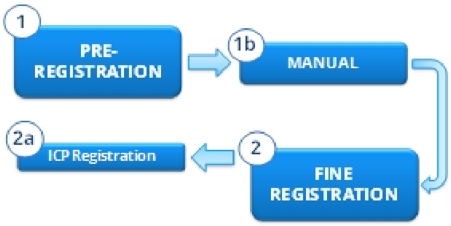
or also a bundle adjustment



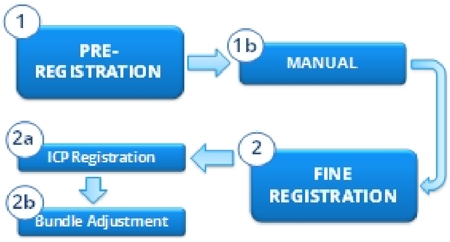
WF 4

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 (only in LineUp® Pro)

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

Target-based

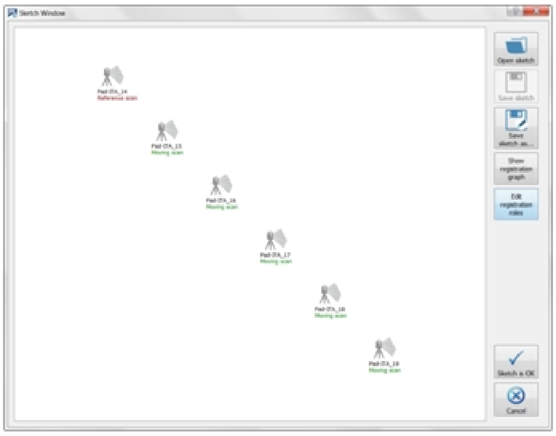
This function automatically registers a set of scans by detecting and matching targets. The user can start the automatic target detection, otherwise can manually add, move and remove targets.

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). If needed, you can register one moving grid to more

reference grids.

Note: in the 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.

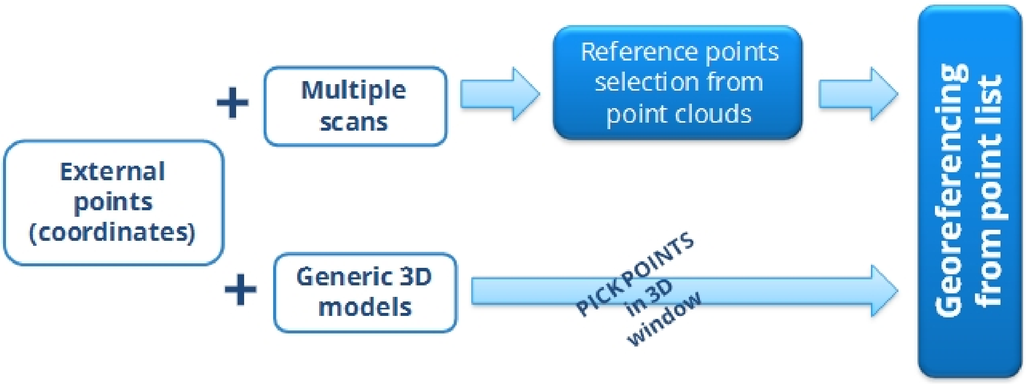
Georeference

The **LineUp®** tool includes a procedure to geo-reference the imported point clouds:

Point clouds georeferencing

Pick reference points

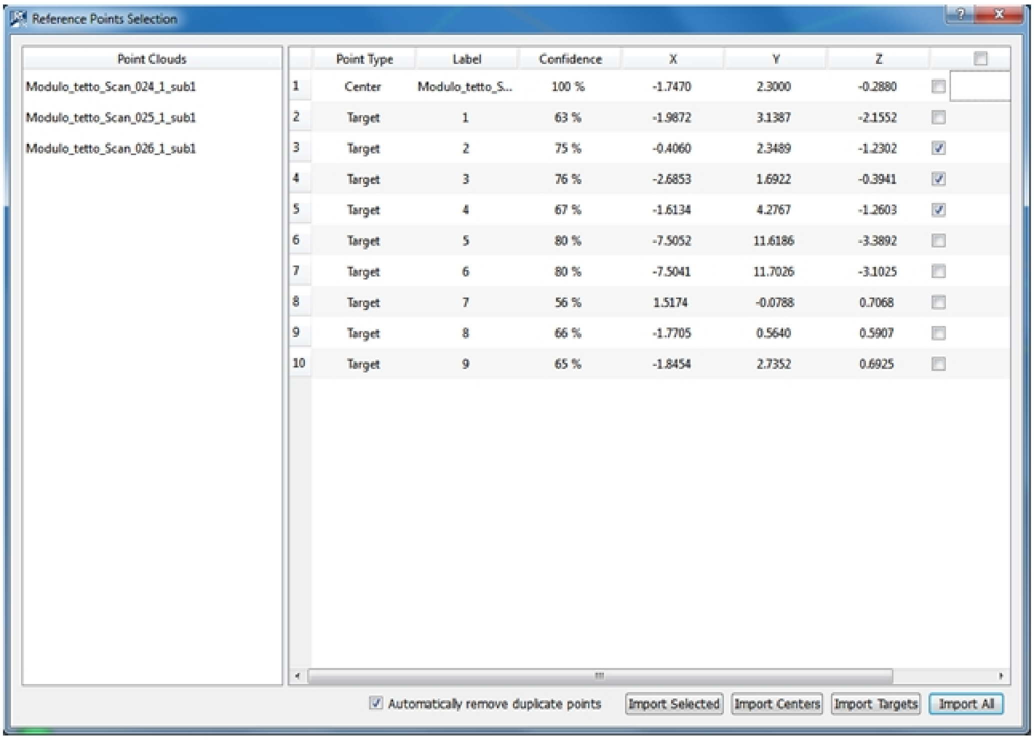
This process allows you to georeference multiple scans using both targets (or other points) and the centers of the scans (if geo-referred).
You can georeference one or more scans by selecting at least 3 points from different scans, known also in another reference system (for example from topography). 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

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 registration dialog or targets editor dialog.



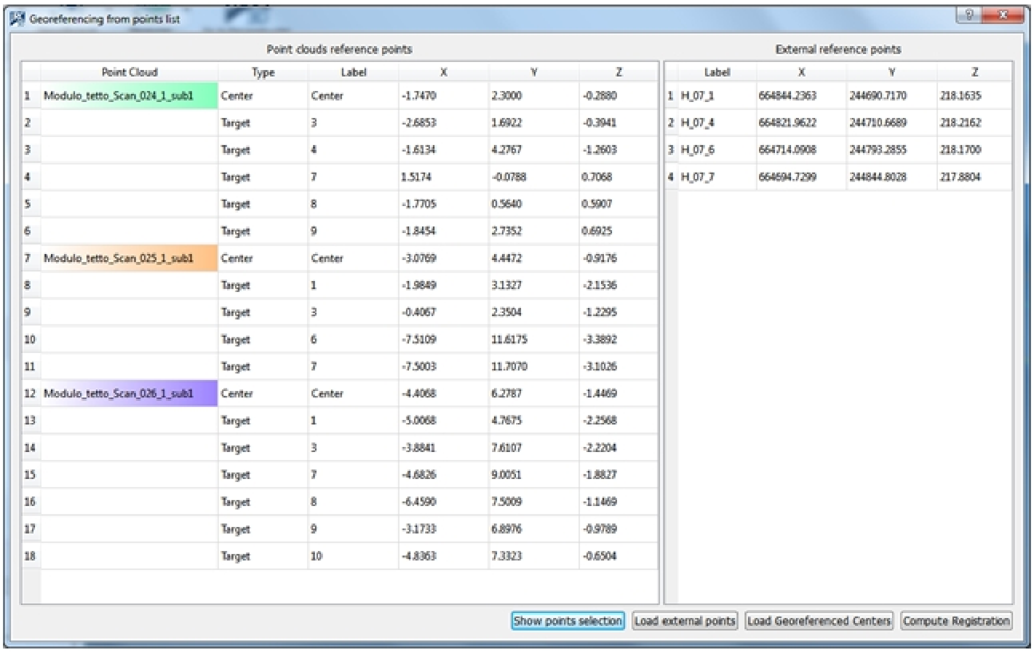
Note
To pick reference points on the point clouds in the project - if you haven't yet made it - you can use the command:

Pick reference points

The reference points taken with this procedure will be automatically loaded in the *Reference Point Selection*.
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 reference points for geo-referencing scans using previous selected target points (or georeferenced scan centers).



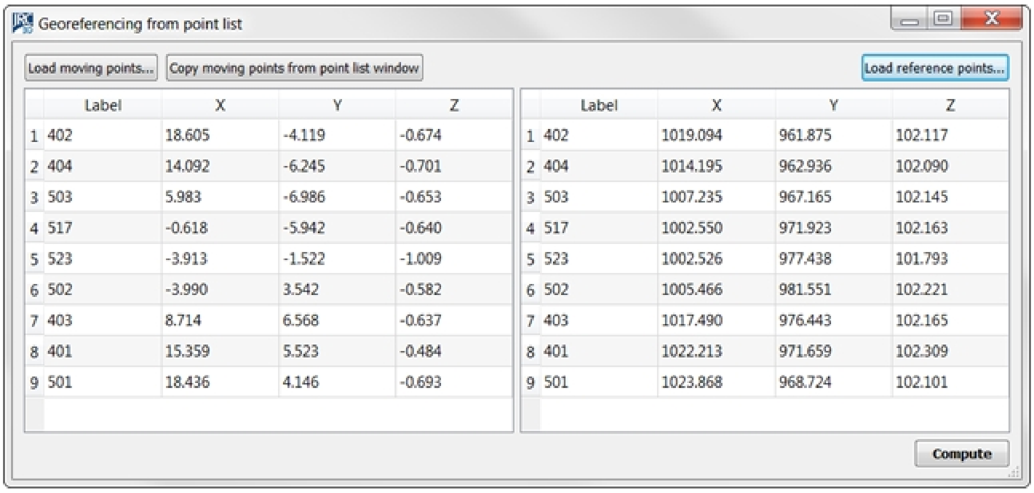
The *Compute Registration* command automatically finds the best matching points from the right and left lists and computes the alignment between them.

3. 3D Model georeferencing

Using the *Tools-> Pose&Registration -> Point list registration* command in JRC 3D 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.



	Label	X	Y	Z
1	402	18.605	-4.119	-0.674
2	404	14.092	-6.245	-0.701
3	503	5.983	-6.986	-0.653
4	517	-0.618	-5.942	-0.640
5	523	-3.913	-1.522	-1.009
6	502	-3.990	3.542	-0.582
7	403	8.714	6.568	-0.637
8	401	15.359	5.523	-0.484
9	501	18.436	4.146	-0.693

	Label	X	Y	Z
1	402	1019.094	961.875	102.117
2	404	1014.195	962.936	102.090
3	503	1007.235	967.165	102.145
4	517	1002.550	971.923	102.163
5	523	1002.526	977.438	101.793
6	502	1005.466	981.551	102.221
7	403	1017.490	976.443	102.165
8	401	1022.213	971.659	102.309
9	501	1023.868	968.724	102.101

Note: to start the registration, the left table has 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).

Edit Point Clouds

In this section you can learn how to manage the point clouds to obtain a better use of the extractable information.

The two main operations that you can apply on the point clouds are:
1. Point filtering and clustering
2. Selection and Fitting

1. 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.
You can access to these procedures by the *Tools-> Points Filtering & Clustering* command or by the point cloud contextual menu.
The filtering tools include also the point cloud Pre-process filters: JRC 3D Reconstructor® applies a set of algorithms to the scans which extract information that is needed during further processing of the data. You can access this command through the point clouds contextual menu, the Tools->Point Filtering&Clustering command and *LineUp Setting Dialog -> Pre-processing dialog*.
All the other commands can be activate through the point clouds contextual menu or the Tools->Point Filtering&Clustering command.

Processes			*On Grid Point Clouds*	*On Unstructured Point Clouds*
Filtering	Pre-process clouds	• Noise Removal (Range & Reflectance Gate, Outlier Removal, Median Filter, Mixed Point Filter) • Compute Normals • Edge Detection (Depth & Orientation Discontinuity) • Compute Confidence	✓	

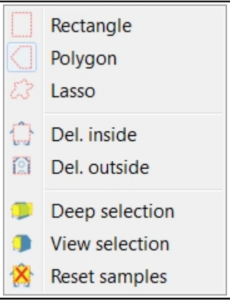
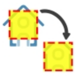
		• Noise Removal (Outlier Removal) • Compute Normals • Compute Confidence		✓
	Restore raw data	To undo any operation of preprocessing, deletion and editing that may have been performed on the clouds.	✓	✓
	Restore deleted points	To undelete all the points earlier deleted.	✓	✓
	Edit 2D	A grid point cloud 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 Point Filtering & Clustering for details.

2. Selection and fitting tools

This set of tools allows you to:

Selection		Select a point clouds portion (to delete or sample it) 
		Make a Point selection with polyline
		Sample to new cloud starting from a Selection
Fitting		Fit a plane from a selection of points
		Fit a cylinder from a selection of points
		Fit a sphere from a selection of points

Create & Edit 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:

1. **Creating meshes**
2. **Editing meshes**

1. Creating meshes

JRC 3D 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

These commands are achievable from *Tools→Meshing*, from *Mesh tools* in the top toolbar or from the context menu of point clouds.

2. Editing meshes

JRC 3D Reconstructor® provides several functions for editing meshes.

In *Tools→Meshing* and/or in the meshes' contextual menu (right mouse button click) you find all the commands able to manage and edit meshes:

Mesh Editor	An editing environment to perform advanced operations such as hole-filling, borders detection, editing triangles and vertexes, smoothing surfaces, decimating, ridges and valleys 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
Make single mesh...	This dialog allows to lump together in a single mesh an arbitrary set of triangle meshes
Convert 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
Compute 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

See the above linked voices for more details.

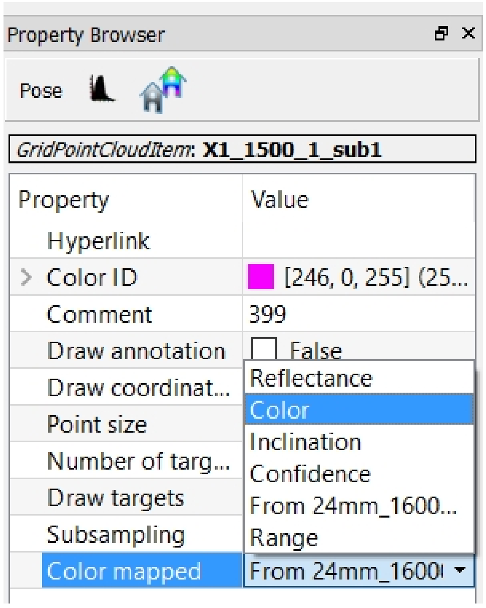
Photo & Color

JRC 3D 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 1.Colors mapping).

The coloring tools are accessible from the *Tools->Photo&Color* menu or from the point clouds' contextual menu *Photo&Color*. See also *Photo and Color* in the Top Toolbar.

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.



See also our video tutorials on point clouds coloring.

1. Colors mapping

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
 ***Colors mapping***	To create 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 point 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. *Suggestion*: use it to classify points for a further vegetation removal.

2. Camera calibration, color point clouds & meshes

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

Create Projector	To create a Perspective, Orthographic, Spherical , Cylindrical projector or from a calibration (by importing a camera calibration file) using a valid image.
Import georeferenced tif	This function imports a geo-referenced TIFF as an orthographic camera.

Main features extraction

Until here you imported, preprocessed, registered and georeferenced your scans, created meshes… to obtain your 3D model.
And now?
Which kind of data you can extract from JRC 3D Reconstructor®?
How you can export them?

Measures & notes
Area&Volume

To take annotations and measures as:

- Distances (linear and angular)
- Area
- Volume
- Compute cut&fill volume

Elevation&Plans (Orthophotos)

To create:

- Orthophoto
- Cameras
- Virtual scan

Cross Sections

To create planes and extract single or multiple cross sections

Inspections

To create 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).

- Analysis deviation
- Planarity&verticality maps

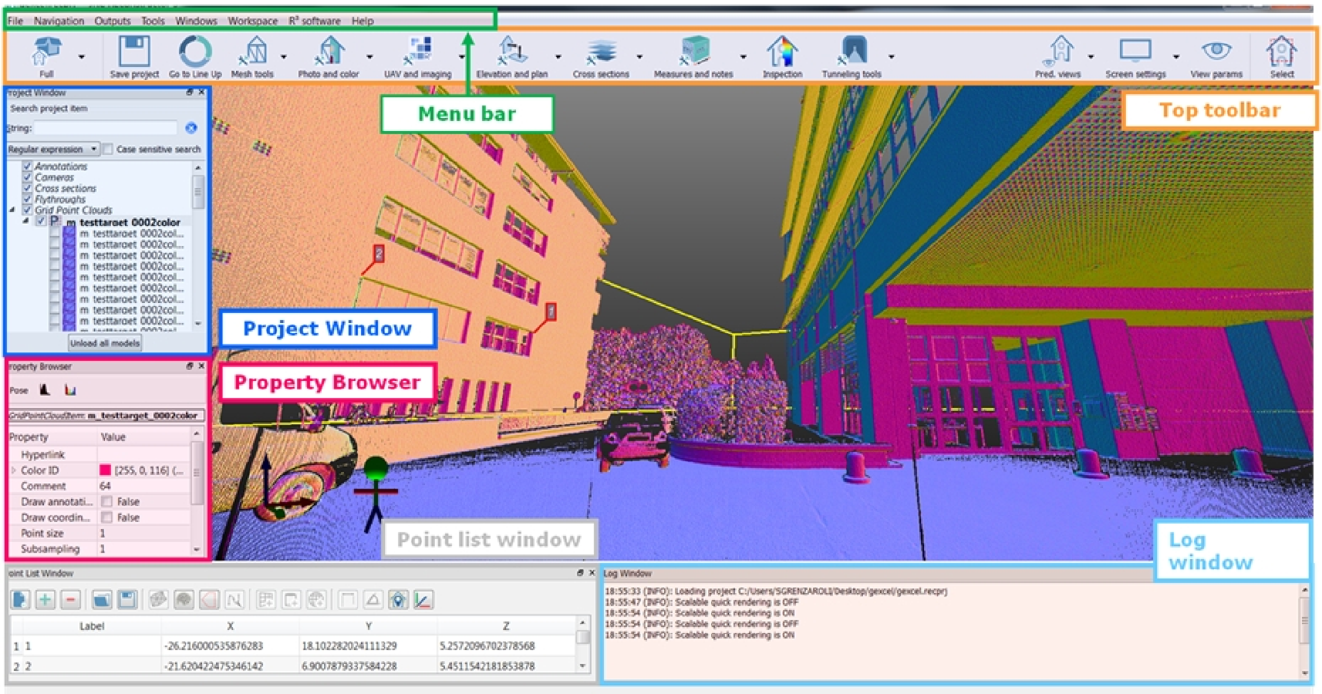
Tunneling

Videos & snapshots

See Outputs menu.

User Interface Layout

The JRC 3D Reconstructor®'s interface is simple and user-friendly, so most of the main commands are in the ribbon Top Toolbar.



The user interface is composed by:

- The **menu bar** on the uppermost part of the window: contains all the command and software functions
- The **Top Toolbar** with shortcuts to the most useful and relevant commands (for each workflow); it's composed of hoverable buttons, just hover with the mouse on any icon of the top toolbar, and the commands of that category will pop down for you to activate. The top toolbar changes when the user changes the current **workspace**
- The **main rendering window**
- The **project window** (dockable), which lists the objects that make the 3D scene (imported or created during the processing)
- The **property editor** (dockable), which allows to view and edit the properties of the selected objects
- Other **dockable windows**

Furthermore, by right-clicking on an item in the project window, a **contextual 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.

Workspaces

A *workspace* is a particular configuration of JRC 3D Reconstructor®'s interface, aimed at satisfying a particular workflow. In this way, top toolbar and dockable windows are reorganized by showing only the functions useful for particular tasks and in a meaningful order and place.
Here you can compare the different workspaces, taking in account the main procedures and tools.

	Workspace	Main features
	***Viewer***	To only navigate the project (and the objects inside it) and take distance measures and annotations. Useful for demo and to show projects to customers.
	***LineUp***	To import, pre-process and register point clouds
	***Survey***	http://gexcel.it/en/software/jrc-3d-reconstructor/special-stonex-reconstructor/survey
	***Mining/Tunneling***	http://gexcel.it/en/software/jrc-3d-reconstructor/special-stonex-reconstructor/mining
	***Construction***	http://gexcel.it/en/software/jrc-3d-reconstructor/special-stonex-reconstructor/construction

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.

Note: the access of each workspace is due to the available licenses!

Menu bar

The menu bar is located on top of JRC 3D 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.

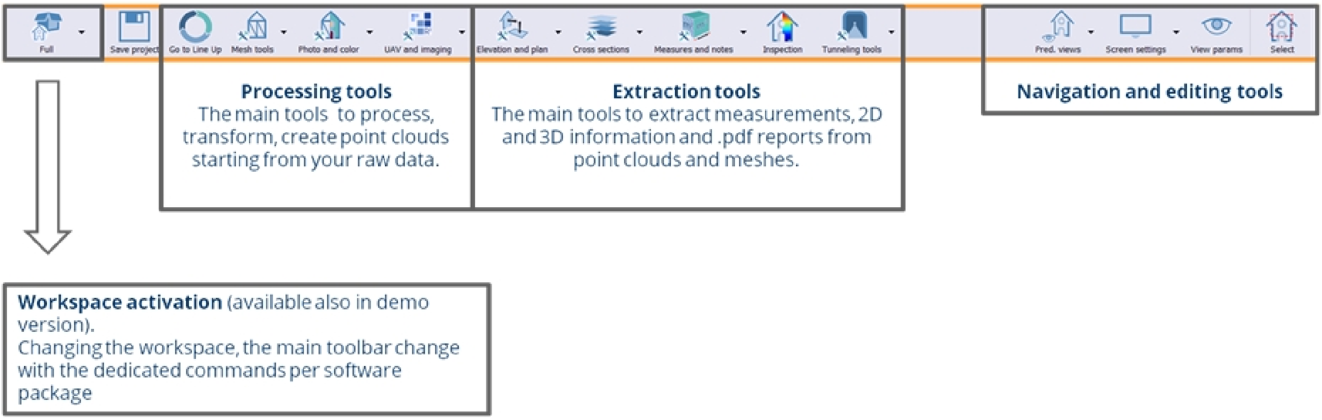
The menu bar features the following main voices:

- File: to import/export, load and save data and to manage the project
- Navigation: to interact with the 3D scene and manage rendering settings
- Outputs: to extract results from the project data
- Tools: auxiliary tools to process the data
- Windows: to manage the dockable windows
- Workspaces: changes the current workspace
- R^3 software: To direct link to R3 software packages installed in your PC
- Help: to access the online help, the license manager and to display information on the software version in use.

See also JRC 3D Reconstructor® user interface.

Top Toolbar

The Top toolbar allows fast access to frequently used commands from the File menu, Navigation menu and Outputs menu.
The toolbar changes heavily when the user changes the current workspace.



Dockable Windows

JRC 3D Reconstructor®'s 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*: its default docking area is on the left side of the GUI
- *Property window:* its default docking area is on the left side
- *Readout window:* its default docking area is on the right side
- *Log window:* its default docking area is on bottom right
- *Point list window:* its default docking area is on bottom left
- *Flythrough editor:* its default docking area is on the right side
- *Manual positioning:* its default docking area is on the right side

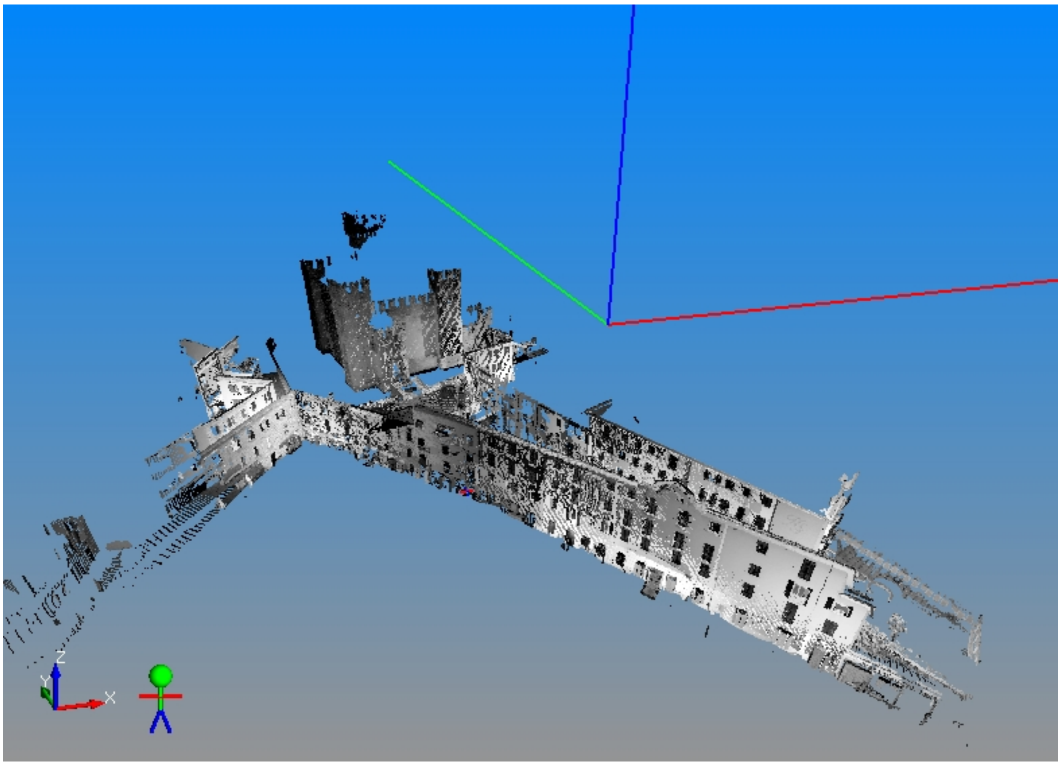
The Recipe Window

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 JRC 3D Reconstructor® at need. It is shown for the time needed by a particular procedure to interact with the user.

Therefore, the recipy 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*

The 3D window



This window is located in the center of JRC 3D 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 files from a Windows Explorer window to the JRC 3D Reconstructor® window.

Navigation

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

Selection

By pressing the space bar, the 3D window goes from *Navigation mode* to *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 coordinate system with
The *X-axis* pointing to the left
The *Y-axis* pointing up and
The *Z-axis* out of the screen

The key concept is the "current camera global reference" (CCGR), i.e. its latitude, longitude and altitude

(LLA). All the items of the current session are rendered with a relative transform of their own global reference from the CCGR. The only way to move the CCGR is going to ("Go to" command) a different item. Until then, the coordinate system is centered in the CCGR, with the XY plane tangent to the earth' surface, X pointing to North, Y pointing to West and Z is the height above the earth, as a right-handed coordinate system. Going far away from the origin of the CCGR the accuracy of the LLA coordinates decreases (shown in the lower right panel), since it's an approximation from the CCGR tangent plane. Thus, if accurate measurement is a concern, better change the CCGR.

Project Items

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

Models

The supported model structures are:
Unstructured Point Clouds: a list of points with no relationship with each other
Grid Point Clouds: a list of points organized in a 2D grid, like an image
Triangle Meshes: a list of meshes, where each mesh is a collection of triangles with adjacency information

Other project items

Polylines: created by contour algorithms or cross sections (exportable in a .dxf file)
Camera views: preferred view points saved by the user with a definable projection among ortho, perspective, cylindrical and spherical
Projectors: images registered in the scene that can be projected on the geometry to create a texture map
Annotations: user comments attached to valid 3D points in the scene. Distance and angle quotations are considered as annotations
Flythroughs: spline trajectories defined by the user in order to create animations and movies
Planes: useful to calculate volumes and cross sections
Projects: useful to merge in a single location all the items of a sub-project.
User Coordinate Systems: the user can create different coordinate systems with special purposes.

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 standard processing functions.

See also project window.

R^3 software menu

This menu allows you to access the JRC 3D Reconstructor® functions needed to run the Gexcel R^3 software:

	Point R^3	To manage and display 3D data and large dataset of 3D point clouds in a very simple way.
	PixR^3	To create 3D models starting from photos. See also UAV and imaging to learn how to integrate JRC 3D Reconstructor and PixR^3.

Note: This menu appears in the Menu bar only if one or more R^3 packages are installed!

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**: open an existing project (*.recprj or *.r2s) • **Save project** • **Save project as**: save a copy of the project with a different name • **Recent projects:** opens one of the project you recently used • **Convert project to efficient format**
IMPORT	• **Import format**: import data from third party formats. • **Import Rec Project** • **Load item**
EXPORT	• **Export** • **Export model as...** • **Export polyline as...** • **Export cross section as...** • **Export UCS to AutoCAD** • **Export annotation to .CSV** • **Export pose**

Exit: quits JRC 3D Reconstructor®. If you have unsaved changes to your projects, JRC 3D Reconstructor® asks you if you wat to save the changes or not before exiting.

Project Management

In this section you can learn how to efficiently manage your JRC 3D Reconstructor® project by using the following commands.

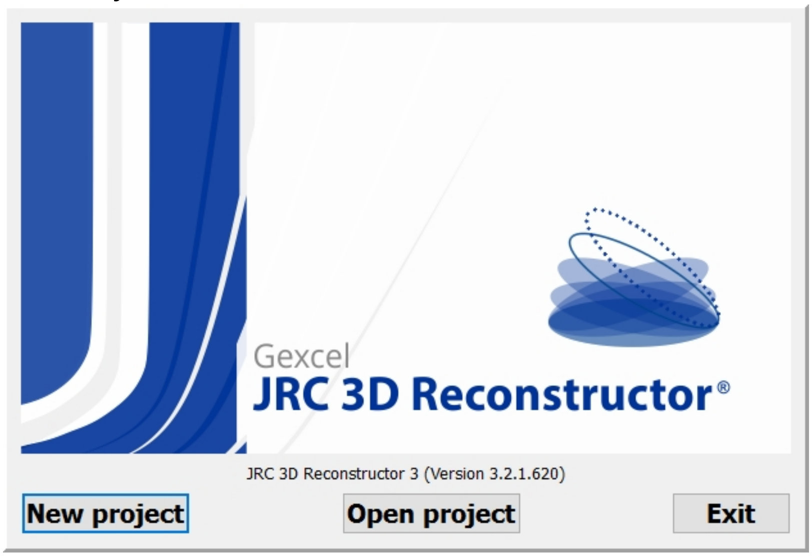
New project

This function allows to create a new JRC 3D Reconstructor® project.
Each project has its own folder called with the same name. JRC 3D Reconstructor® projects contains at least some of the following subfolders:

- *Imports/*
- *Exports/*
- *Unstructs/*
- *Grids/*
- *Meshes/*
- *Polylines/*
- *Trash/*
- *Images/*
- *Movies/*

These folders are automatically created when the project gets populated.

The option to create a new project appears also in JRC 3D Reconstructor®'s initial screen, shown below, and has the same functionality.



Open project

This function opens a JRC 3D Reconstructor® project (*.recprj, *.r2s). Before opening a new project, JRC 3D Reconstructor® asks whether you want to save the current one (if you left unsaved changes).

Compatibility across different JRC 3D Reconstructor® versions

JRC 3D Reconstructor® projects are backwards compatible. For JRC 3D 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, JRC 3D Reconstructor® 3.0 can open projects saved with versions 2.x; however it cannot open projects saved with JRC 3D Reconstructor® 3.1. However, JRC 3D Reconstructor® 3.0.0 can open projects saved with JRC 3D Reconstructor® 3.0.1.

Persistence of camera pose

When opening a project, JRC 3D 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, 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.

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.

Recent projects

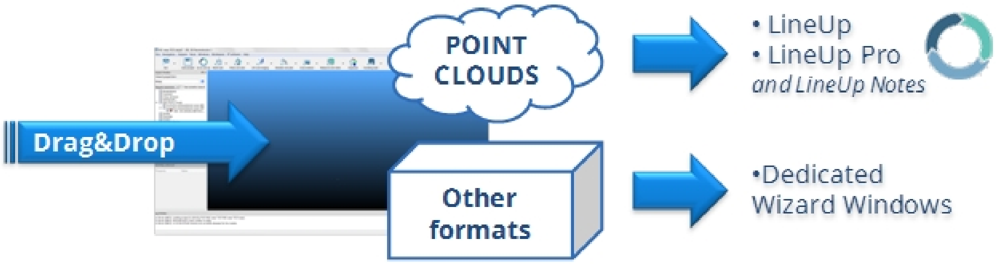
This function allows you to quickly open one of the projects you recently worked with. JRC 3D Reconstructor® keeps track of the latest 20 projects used in this menu option.

Convert project to efficient format

This function converts all data of the current project in the new efficient format, that guarantees faster reading and writing and better compression ration.
Since JRC 3D Reconstructor® v. 3.0, all new projects are already written in the efficient formats. Therefore, this function is only useful to convert projects that were created with earlier Reconstructor® versions.
If you have many old projects to convert, or whole external HDDs with projects to convert, you may want to use the R2ConvertProjects.exe tool present in JRC 3D Reconstructor® installation folder. Just drag and drop the root folder of your drive into that executable. R2ConvertProjects will search recursively for all subfolders containing Reconstructor® projects and convert them to the 3.0 efficient format. Your point clouds and meshes will be read and written faster, and you will get back many gigabytes of free space.

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 kink of data.



a. If you're importing a point cloud the **LineUp® Tool** 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.

- **Import Format**
- **Import Rec Project**
- **Load Item**

See also File menu.

Import Format

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 plugin 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.

Grid Point Cloud

Comma Separated Values .csv
DEM ASCII header+matrix .asc
Generic binary raster *.*
Grid Point Cloud Text File .txt
PTX .ptx
Stonex multiple scans .x3m
Stonexpoint cloud .x3s

Mesh

3DS .3ds
AutoCAD data exchange format .dxf
Collada .dae
IFC (Industrial Foundation Classes) .ifc
PLY mesh .ply
Reconstructor VG3 mesh .vg3
STL Mesh .stl
Triangle Meshes Text file .txt
VRML .wrl, .vrml
Wavefront .obj

Unstructured Point Cloud

E57 unstructured point cloud .e57
Kubit PointCloud .ptc
LAS format .las
PLY point cloud .ply
Unstructured Point Cloud Text File .txt
PTS .pts

Polyline/trajectory

AutoCAD data exchange format (*.dxf)
List of points (*.txt)

CAD Models

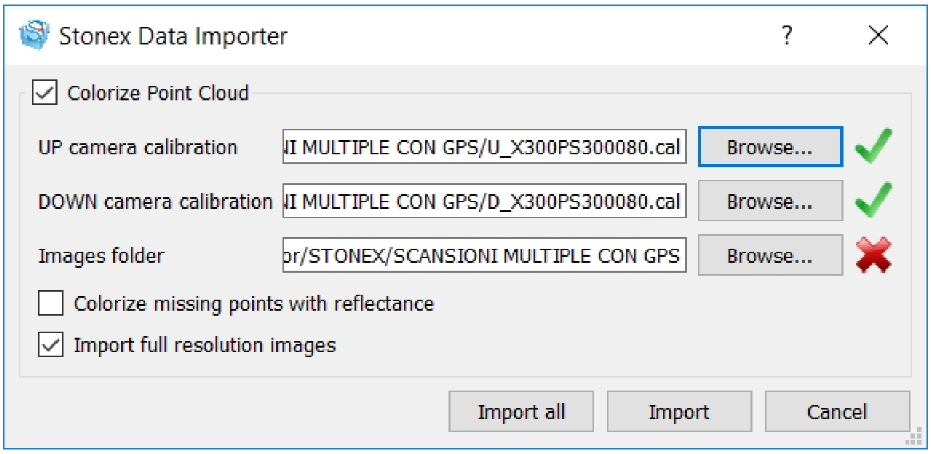
AutoCAD data exchange format .dxf
IFC (Industrial Foundation Classes) .ifc
STL Mesh .stl
VRML .wrl, .vrml

See also the Stonex Data Importer.

Stonex Data Importer

Stonex Reconstructor is able to import the Stonex Cloud files automatically colorized by using pictures and calibration files.

During the import process the following wizard



help you to:

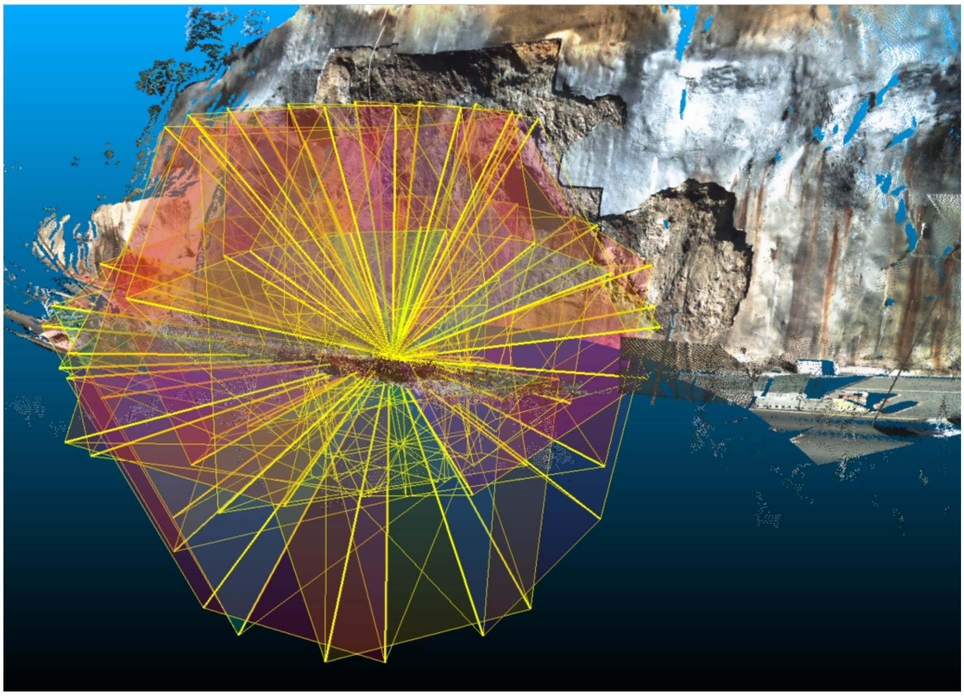
1. *Colorize the Point Cloud*, importing UP/DOWN camera calibrations and images.

   You need to load:
   - .x3s and .x3m scans files
   - pictures of UP (U) and DOWN (D) cameras (. jpeg)
   - calibration of UP (U) and DOWN (D) cameras (. cal)

   It's suggested to keep these files in the same folder.

2. *Colorize missing points with reflectance*, to colorize points with reflectance where color information doesn't exist.

3. Import full resolution images: to import also projectors of pictures.



By clicking on Import all, the prevoius choises will be applied to all the scans you're importing.

Import CAD models

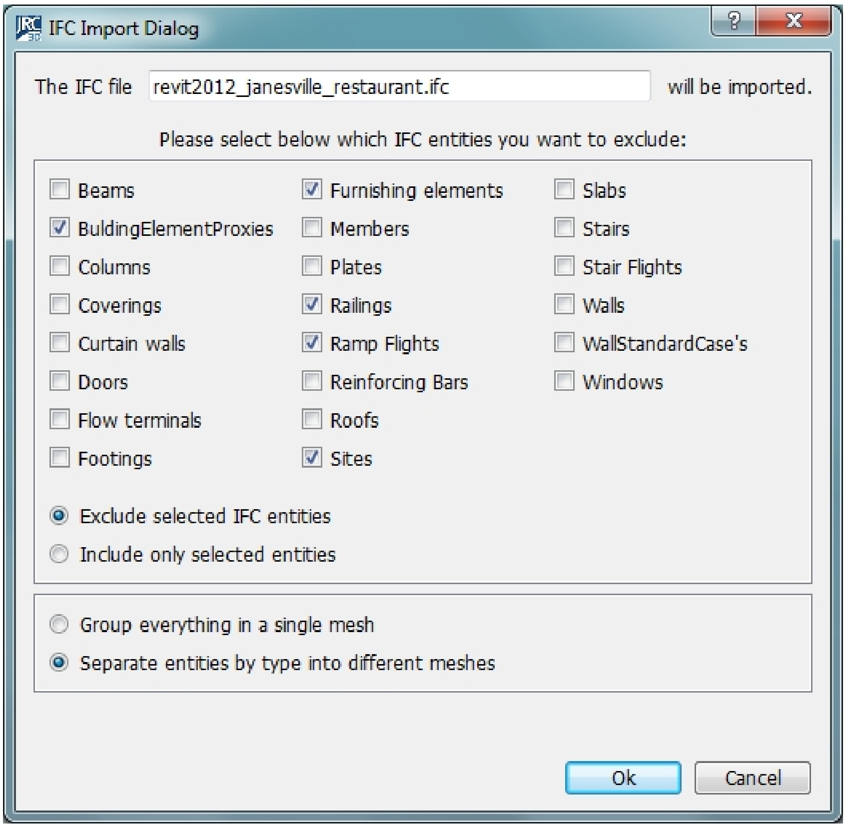
This function allows you to import in JRC 3D Reconstructor® models from CAD softwares, in order to do structural analyses or to compare the models with the point clouds.

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, JRC 3D 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. JRC 3D Reconstructor® preserves the model colors (except for the transparency)
STL Mesh .stl	When importing STL files, JRC 3D 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 JRC 3D 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, JRC 3D Reconstructor® will maintain their colors and textures. JRC 3D 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 JRC 3D 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 JRC 3D Reconstructor®. The above dialog opens up.

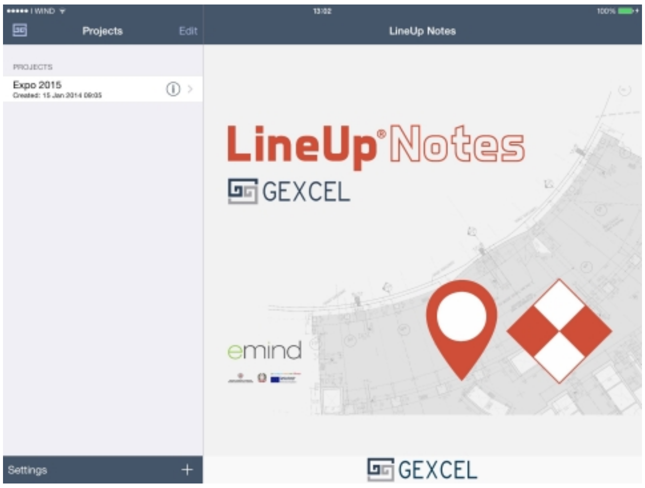
Exclude unneeded entities

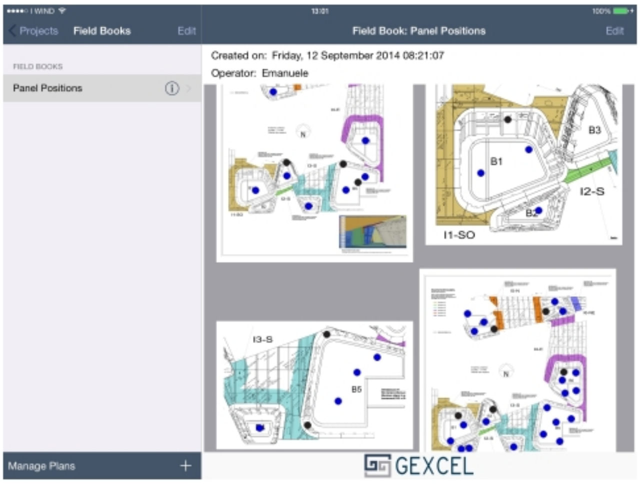
BIM projects can be very large and detailed, including furniture elements, railings, windows, etc. For the sake of inspection, you may want to exclude some project entities because, for example, they are simply not yet in place in reality. To exclude some entity types, just mark them among the check 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, JRC 3D 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.

Import sketch from LineUp® Notes





LineUp® Notes is a Gexcel APP dedicated to:

- *Import plan views of your site*: Easily manage large scanning projects and speed up the global scans/point clouds registration. When in few days a huge number of scans are taken by different surveyors in different positions. Plan your work in advance, or while you are surveying, and optimize it.
- *Note scan/targets position and properties:* For each scan you can take a note of useful characteristics or personal data of the surveyors and insert your comments. You can insert also more notes like scanning timing, scanner model (by choosing it among the most popular ones: Focus3D, Stonex and Z+F) and all the notes you think can be necessary.
- *Seamlessly, automatically process your data*: Easily connect iPad and PC, to import scans positions and properties in **JRC 3D Reconstructor®**, and seamlessly run automatic, target-less registration using **LineUp® Pro**. LineUp Notes also organizes your survey projects for efficient post-processing and sharing.
- Connect your survey positions and data with your PC using JRC 3D Reconstructor®, to easily organize your elaboration and seamlessly run the automatic target-less registration process using LineUp® Pro.

System Requirements:

- iOS 8.0 or later
- Compatible with iPad mini and iPad retina

You can learn how to:

- install the APP here
- use the APP and import sketch in JRC 3D Reconstructor® here.

You can free download LineUp® Notes here!

Import Rec Project

These function imports items from another JRC 3D Reconstructor® project. The items data can be copied or moved from the project to be imported.

See also UAV and Imaging.

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 JRC 3D Reconstructor® internal formats for items.

Export

This menu contains a number of options to export JRC 3D 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 annotation to .CSV
- Export pose
- Export to Autodesk ReCap 360

Click here to browse the complete list of export formats supported by JRC 3D Reconstructor®. When exporting models or other entities, the coordinates are exported as they appear in the current UCS.

See also File menu.

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

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, JRC 3D 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 unstructured point cloud	Choosing local or global coordinates. Choosing whether to export a color layer and which.	
Generic binary raster *.*		
Grid Point Cloud .rgp		
Grid Point Cloud Text File .txt		
PTC Kubit Point Cloud (unstructured)	Selecting which color layer to export. Setting a subsampling factor for the exported cloud. Skipping the depth discontinuities or not. Skipping the orientation discontinuities or not	
LAS format point cloud (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.	
Optech IXF scan (structured)		
PLY point cloud (unstructured)	Selecting whether to export a color layer and which. 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.
PTX point cloud (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.	

Unstructured Point Clouds

Export format	**Supported options**	**Limitations**
E57 unstructured point cloud	Choosing local or global coordinates. Choosing whether to export a color layer and which.	
Grid Point Cloud Text File .txt		
PTC Kubit Point Cloud (unstructured)	Selecting which color layer to export. Setting a subsampling factor for the exported cloud. Skipping the depth discontinuities or not. Skipping the orientation discontinuities or not	
LAS format point cloud (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.	
PLY point cloud (unstructured)	Selecting whether to export a color layer and which. 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.
Unstructured Point Cloud Text File .rup		
Unstructured Point Cloud Text File .txt		
PTS point cloud .pts		

Meshes

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.
Reconstructor VG3 mesh .vg3		
STL mesh		Does not support mesh color or mesh textures. Only local coordinates are saved, the model global positioning is lost.
PLY mesh	Selecting whether to export a color layer and which. 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), JRC 3D Reconstructor® will not export it in UCS coordinates, because of PLY format limitations. Does not support mesh color or mesh textures.
Triangle Meshes .rtm		
Triangle Meshes Text File .txt		
VRML .wrl, .vrml	Supports mesh textures.	
Wavefront .obj		

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.

Sample script created by this function

```
;JRC Reconstructor UCS creation into ACAD
;----------------------------------------script
_ucs _w
_ucs _na _d UCS_from_Site7-Faro_sub1
_ucs          _3         1.2316973441870025,-12.792058116493525,0.09258080613814651
2.2311556440494402,-12.759596396830426,0.087163833496514284   1.1992797323578062,-
11.792616050403021,0.10062163768949305
_ucs _na _s UCS_from_Site7-Faro_sub1
_plan _c
_ucs _w
_ucs _na _d UCS_from_Site7-Faro_sub1
_ucs          _3         1.2316973441870025,-12.792058116493525,0.09258080613814651
2.2311556440494402,-12.759596396830426,0.087163833496514284   1.1992797323578062,-
11.792616050403021,0.10062163768949305
_ucs _na _s UCS_from_Site7-Faro_sub1
_plan _c
;-------------------------------------endscript
;
;GEXCEL
```

Export annotations to CSV

This function exports a set of point annotations obtained with the annotation tool. Select a list of annotations from the project window, and select this function to export them in .CSV format. JRC 3D Reconstructor® will ask you for a filename and create a .CSV file that you can open in Microsoft Excel or other spreadsheet programs. In the .CSV file, your annotation data will be organized in six columns, and as many rows as the annotations are. The first column will contain the annotations' unique IDs. The columns from second to fourth will contain the annotations' cartesian coordinates. The fifth column contains the comments, and the sixth and last will host the annotations' hyperlinks, if any.

Export pose

This is a very useful function for registration workflows. It allows to safely store in a separate folder the 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 *Exports/* project 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 360 Pro

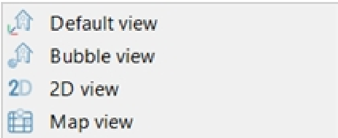
Thanks to free Gexcel ReCap plug-in, JRC 3D Reconstructor® can directly export in .rcs and .rcp formats for a more complete and faster integration with the Autodesk® Recap 360 Pro software. JRC 3D Reconstructor point clouds (.rgp, .rup) can also be directly imported in ReCap 360 Pro.

You can download the free plugin here.

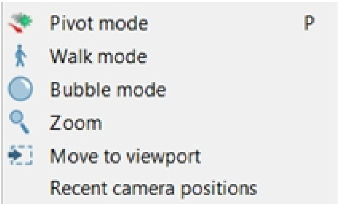
See the videotutorial to learn how to install the plugin and work with it.

Navigation menu

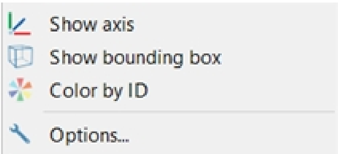
JRC 3D Reconstructor® provides many interaction styles to navigate in the 3D scene.
The navigation modes are selectable through the *Navigation toolbar* (placed on the right) or by *Navigation* command in the top menu.



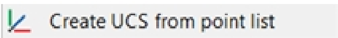
Predefined views



Navigation modes



Screen settings



Creation of User Coordinate System

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.

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

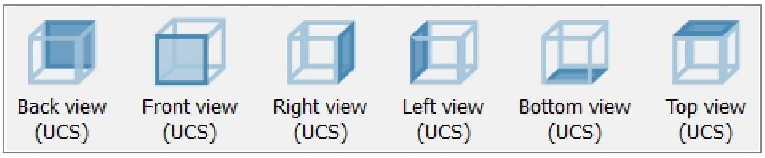
This navigation mode opens the Edit 2D Grid Window. Note that this view is supported for grid point clouds only.

Map view

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

UCS views

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



See also here.

In the **LineUp®** tool other predefined views are available:

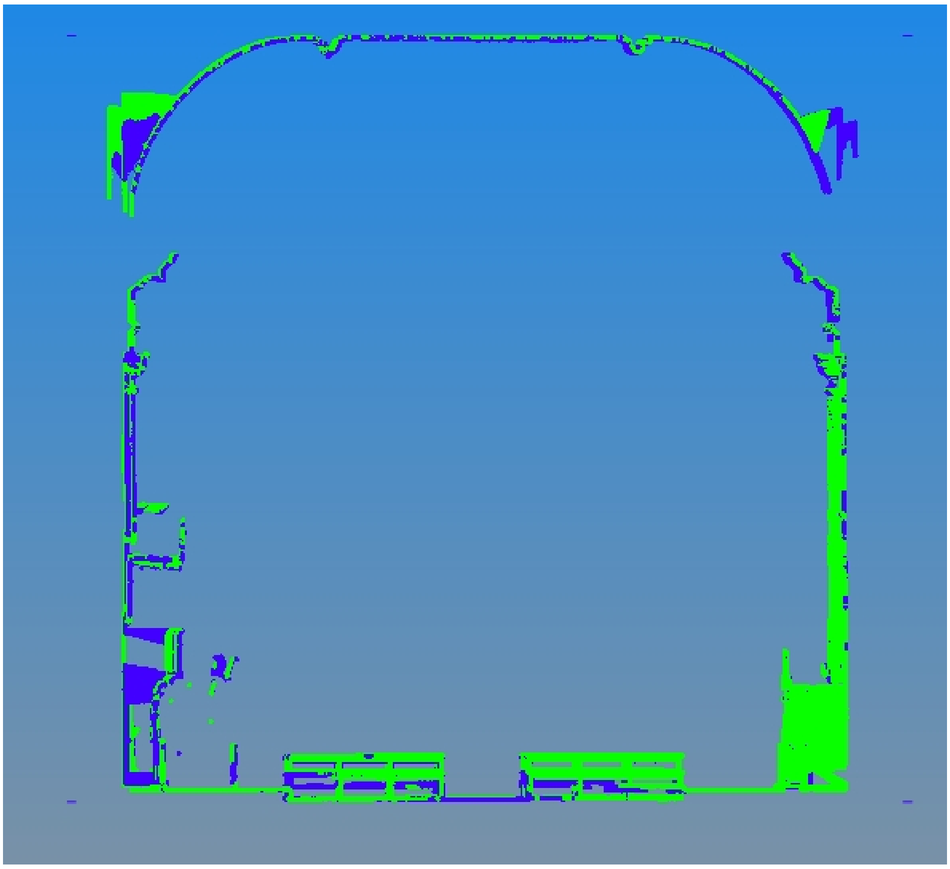
- Slice View
- Horizontal Section View

These modes can be used to check the correct alignment along a vertical/horizontal section of the displayed items.

Slice View (Line Up)

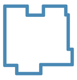
This navigation mode automatically switches the view to orthographic mode and alignes it to the selected items bounding box vertical plane.

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 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

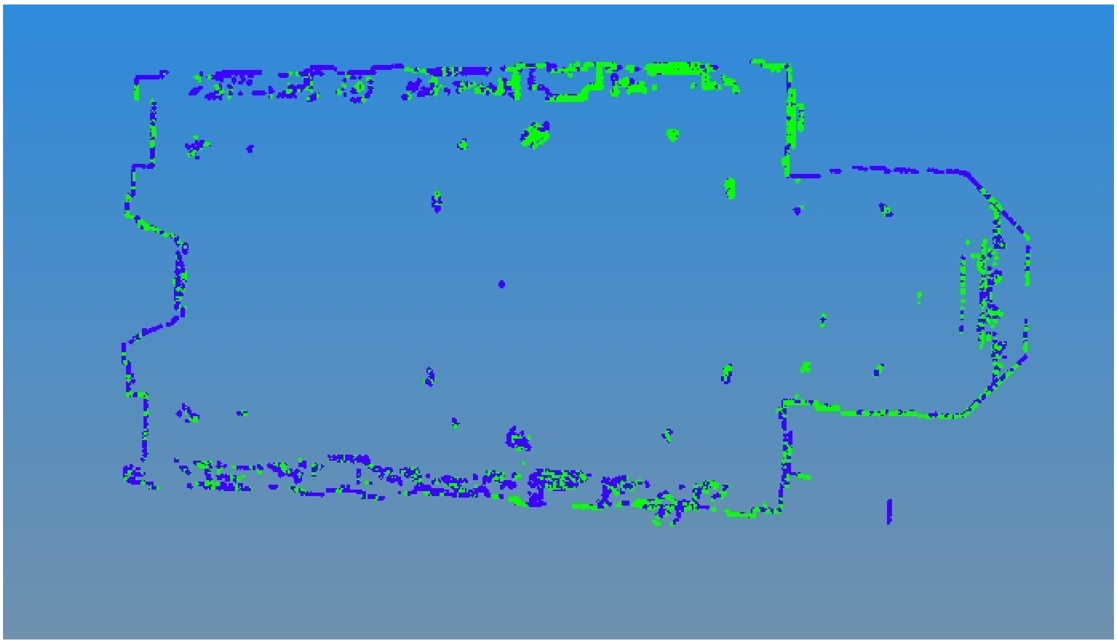
Horizontal Section View (Line Up)



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 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

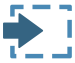
Navigation modes

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

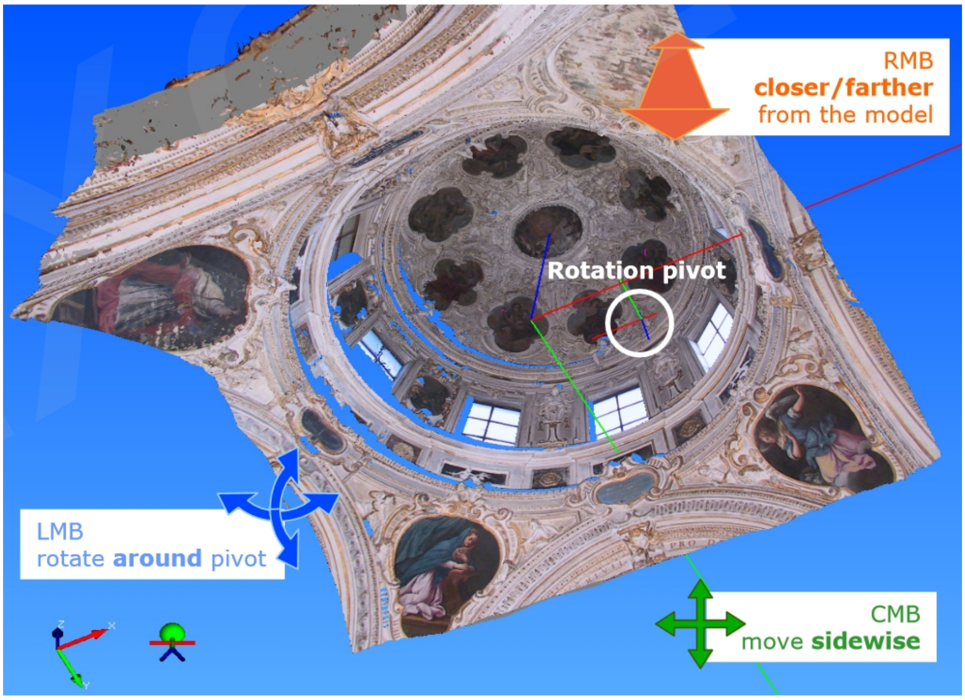


Move to viewport
Moves the virtual camera so that a particular viewport is in sight, without changing the field of view

Recent camera positions
Recovers up to 1000 steps you went through during your navigation

Pivot Mode

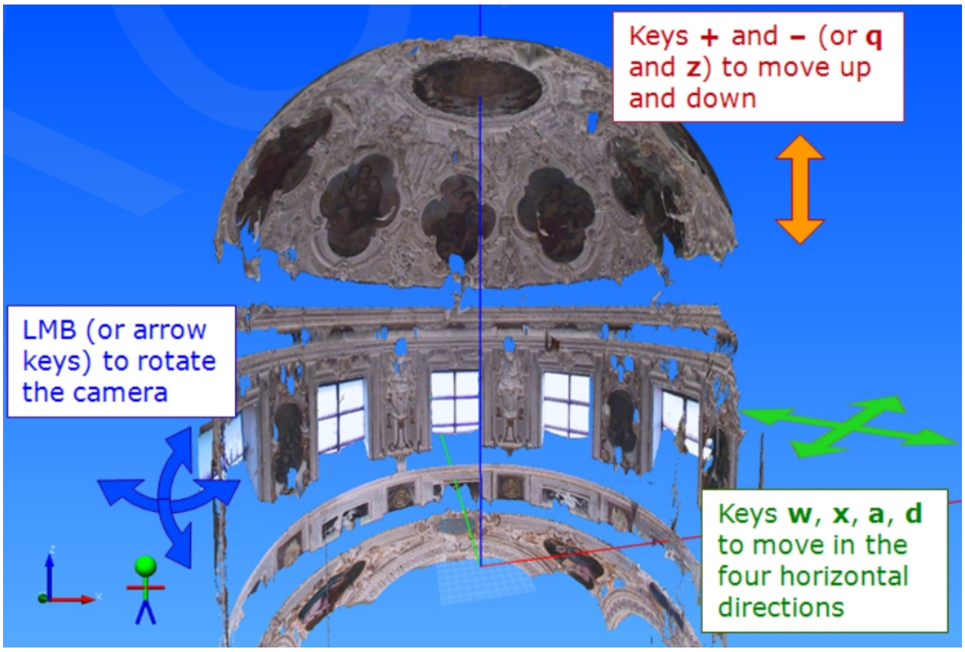
In this mode a point can be selected as a rotation pivot. *The pivot is set by a left button click on a valid point*.



Keyboard/mouse movement	Effect
LMB + move mouse	Camera rotation (pan and tilt)
CMB + move mouse	Camera sideways translation (tracking and booming)
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
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
Key 'c'	Copy readout window's contents to clipboard

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”.

JRC 3D 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.

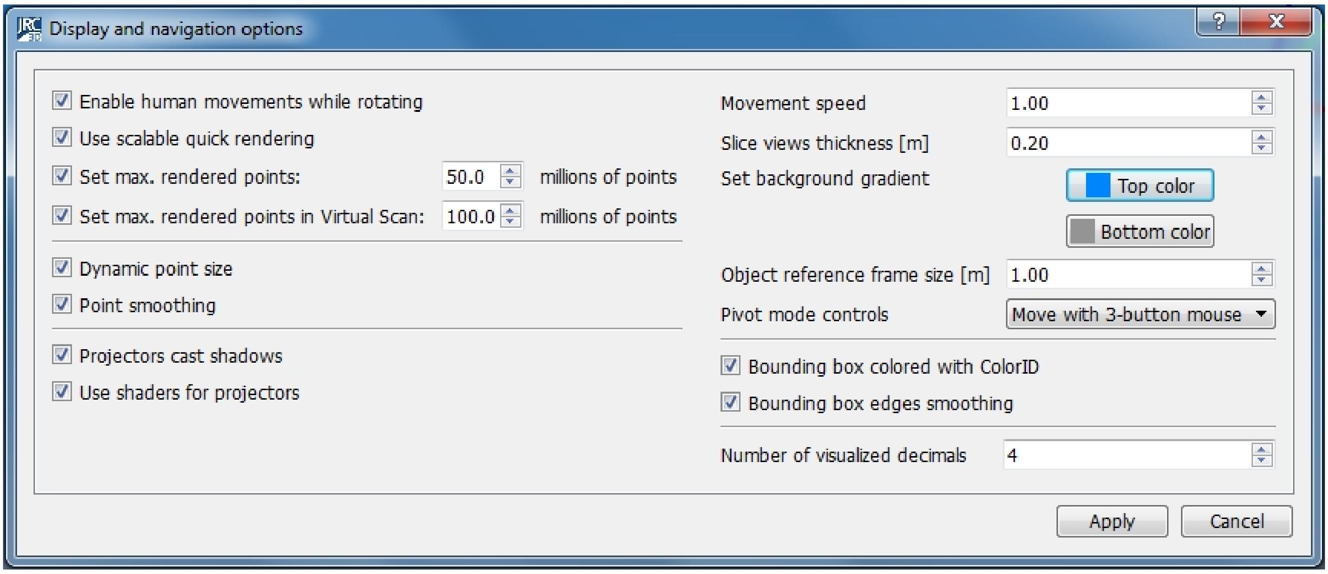
Color by ID

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.

Options

To modify current navigation and rendering parameters.
See details here.

Display and navigation options



This dialog is accessible from the Navigation menu, by selecting *Navigation->Options* or from the *Screen settings -> Options* top toolbar menu.The options present in this dialog are described below.

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.

Use scalable quick rendering

When this option is checked, the *scalable quick rendering* is on. When hundreds of millions of points must be rendered, this rendering system allows you to still navigate very fast across your models. This is possible by rendering only 500000 representative points of all the loaded point clouds. After one second that you have stoped navigating, the rendering of the complete set of points will start.
If this option is unchecked, JRC 3D Reconstructor® will render the complete 3D scene, with up to the defined maxmimum rendered points, for each refresh of the 3D window. In selection mode, the scalable quick rendering is automatically disabled.

Set maximum rendered points

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.

Set maximum rendered points in Virtual Scan

This option is used to set a maximum number of point to be rendered in the Virtual Scan tool. The limit is cumulative for all the loaded point clouds.

Dynamic point size

If this option is enabled, the size of the rendered 3D points decreases linearly with the increasing distance of the points to the “virtual eye”. If this option is disabled, the size of the rendered 3D points is constant regardless of the distance from the viewing point.

Point smoothing

Enable this option to render 3D points as small circles, disable it to render 3D points as small squares.

Projectors cast shadows

If the graphics card supports shadow mapping, this option allows to render projectors that cast shadows through the scene geometry. For best quality, render only triangulated models. Although this can be done also with point clouds by increasing the point size, color projected by the projector could “bleed through” the occluders.

Use shaders for projectors

If the graphics card supports CG shaders, this option can be enabled, which allows to render projectors with the highest quality. Nvidia graphics cards always support CG shaders, while ATI graphics cards don't.

Movement speed

This option sets the translation speed when panning in pivot mode.

Slice views thickness

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

Set gradient background

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 the property. 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 "*Move with 3-button mouse*". These options are two sets of controls for the Pivot Mode. The second one is the default one 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 centre 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.

Bounding box edge smoothing

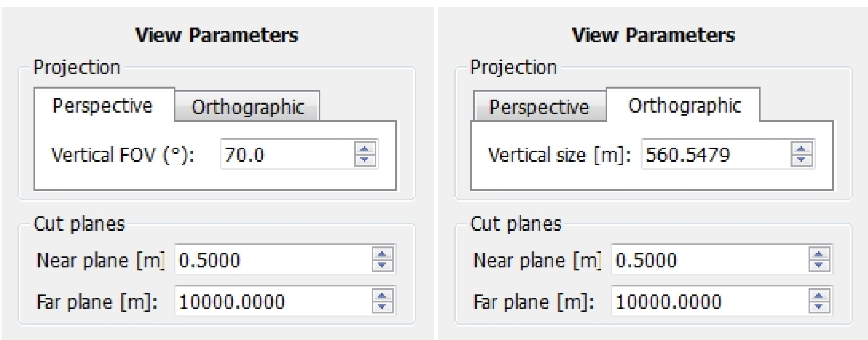
Apply an anti-aliasing algorithm to bounding box edges. It is suggested to leave this option checked unless the graphic card does not support lines smoothing.

Number of visualized decimals

This option sets the number of decimals do be displayed for measures in JRC 3D Reconstructor®. A restart of the software is needed in order for this option to take effect.
Please note that this option only affects the visualized values, while the internal computations are carried out using the full precision.

View Parameters Dialog

This window appears when the user hovers the mouse on the *View parameters* button in the top toolbar. 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 photocamera.

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.

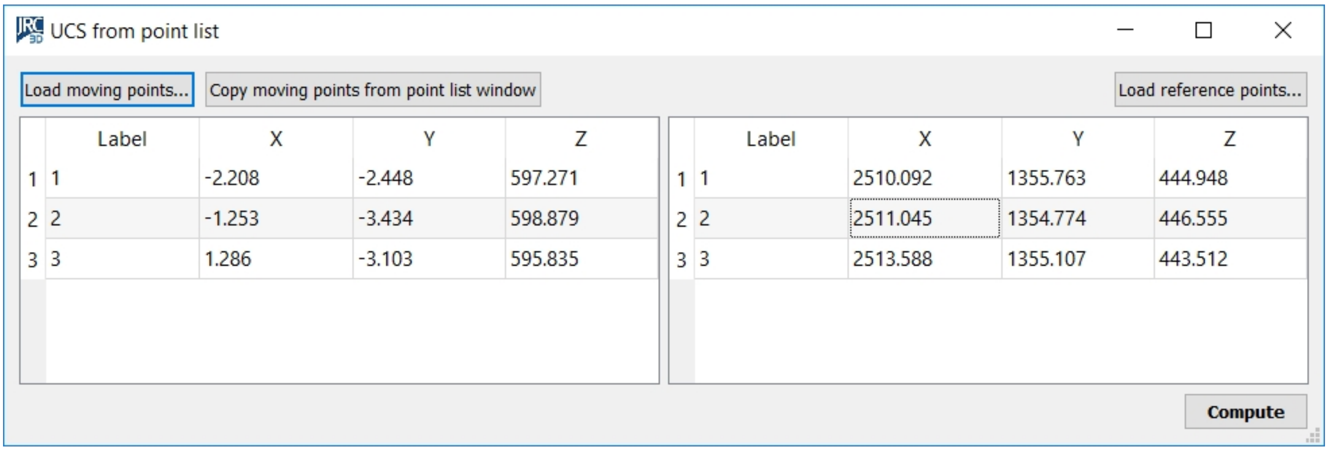
Cut planes

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.

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.

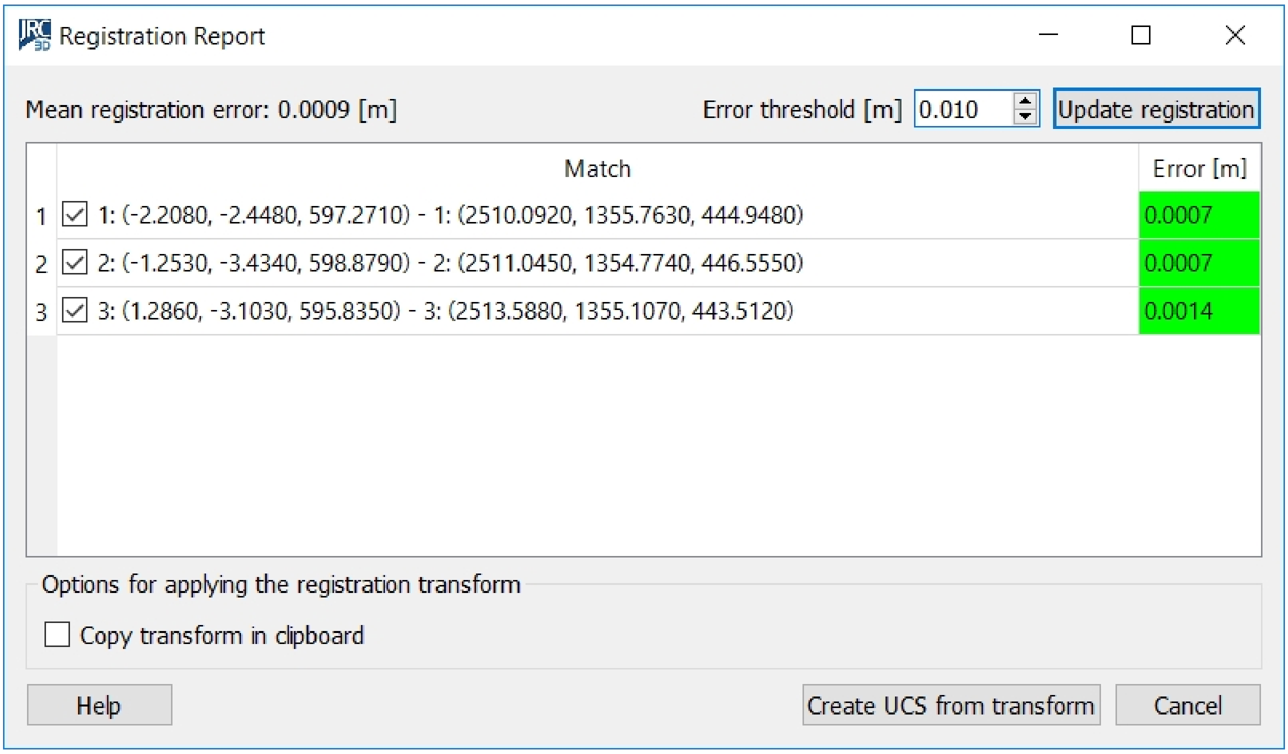


This function is accessible from the Navigation menu, or from the toolbar of the point list window.

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.

Outputs menu

This menu contains many functions to extract results from the project's data and models. These functions are organized in the following submenus:

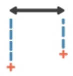
- Measures & notes
- Elevations & plans
- Cross sections
- Areas & Volumes
- Inspection
- Tunnel survey
- Video record
- Drawing tools
- Save snapshot
- Manage units of measure

Measures & Notes

In JRC 3D Reconstructor® some basic measure feature is 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.



Distance

This button allows you to measure a distance between any two points in the 3D scene. After activating this button, press the Left Mouse Button of the mouse (LMB) for the first point and keep it pressed while dragging the mouse to the second point, then release the LMB. A result dialog will pop up displaying the distance between the two points. You will be offered the option to save the distance as an annotation in your project.

Angle

This button enables you to measure and angle between three points in your 3D scene. After activating this button, click the mouse's left button (LMB) for each of the 3 points (must be valid 3D points). The angle of the 2nd point from the 1st and 3rd is computed, and a result dialog opens up to display the result in degrees. You are also presented with the option to save the angle measure as an annotation in your project.

Annotation

This function allows you to create annotation 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 and appears also in the Annotations folder of your project. You can edit the annotation's comment and set an associated hyperlink in the Property Editor.

Exporting meaningful points to Excel or other softwares
Using this function, you can annotate the meaningful points of your project (e.g. topographic targets) and save these annotation as project items. Successively, you can export in .csv these 3D annotations and import them in Excel or other external tools for further processing.

See also Basic measures menu.

See also Areas&Volumes.

Elevations and plans

This submenu contains all the commands and features to create precise, high-resolution elevations and plans of your models:

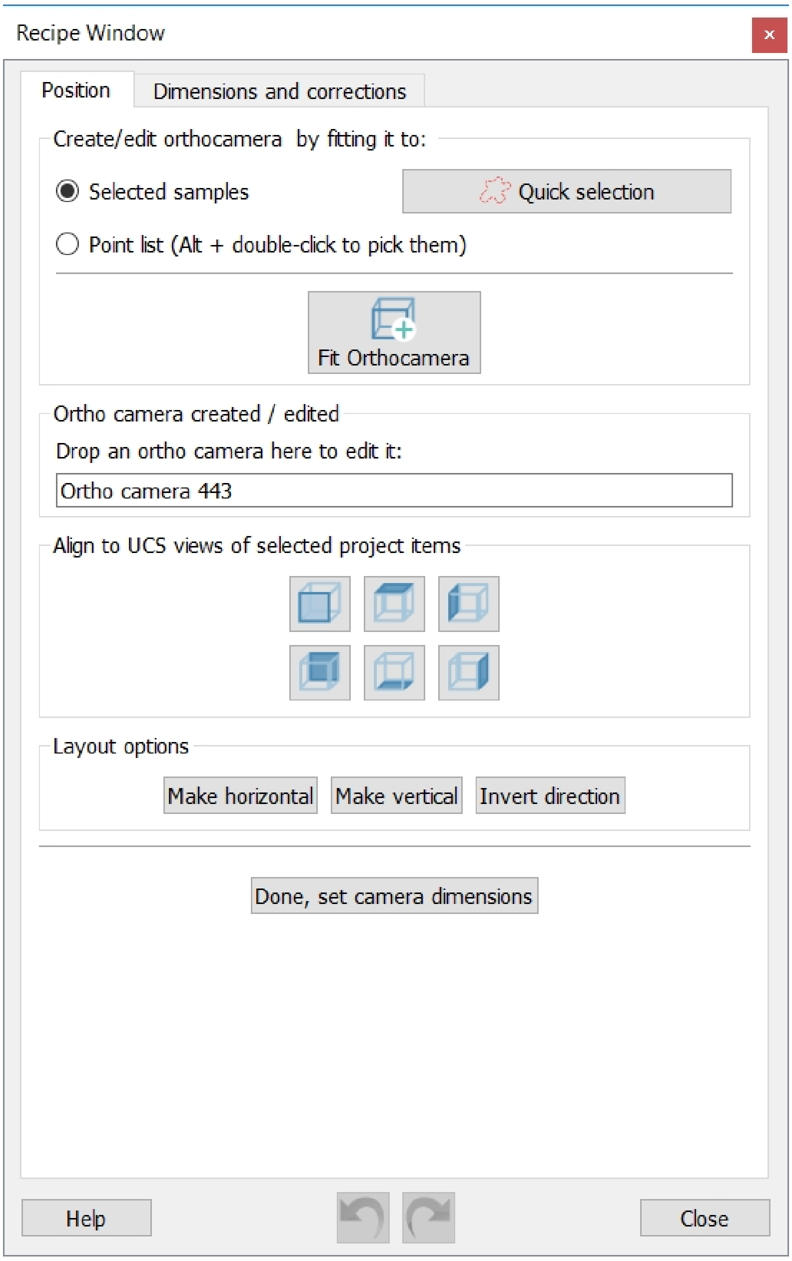
- Create Orthophoto
- Create Camera
- Place here survey point
- Virtual scan

Create Orthophoto

With this toolbox you can create an orthocamera from samples, points, axes, UCS directions, etc., edit it and extract an orthophoto through the Virtual Scan tool.

The recipe window is splitted in two panels:

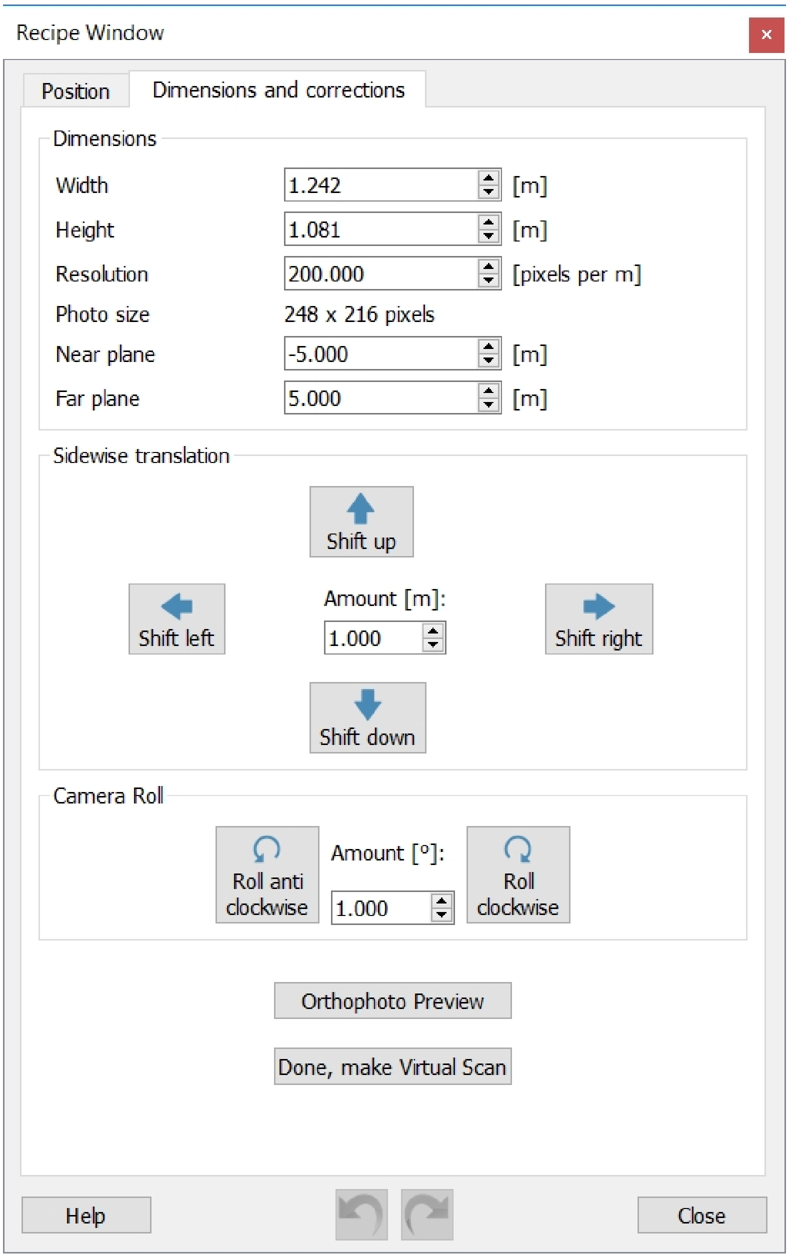
- Position
- Dimensions and corrections



Position

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 Orthocamera button an orthocamera is created. 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 *Done, set camera dimension* button to swith in the *Dimension and Corrections* panel.



Dimensions and corrections

Here you can set the properties of the camera: dimensions, resolution, near and far plane distance.
By using the shift command you can translate and rotate the camera in the space, as you see it in the 3D window.

Orthophoto Preview button 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 *Done, make Virtual scan* button to enter in the *Virtual Scan* window.

See the following videotutorial to learn how to create an orthophoto with an example:

- Orthographic View Extraction Floor
- Orthographic View Extraction Façade

See also Virtual scan andExporting orthophotos to AutoCAD.

Create Camera

This submenu contains functions to create

 prespective

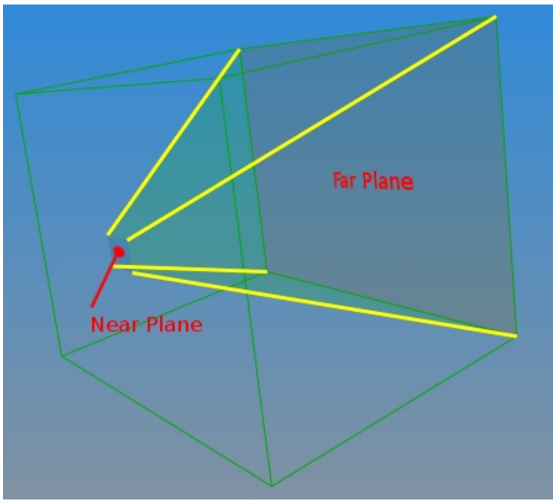
 orthographic

 cylindrical

 spherical

cameras with origin in the current point view.

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 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.

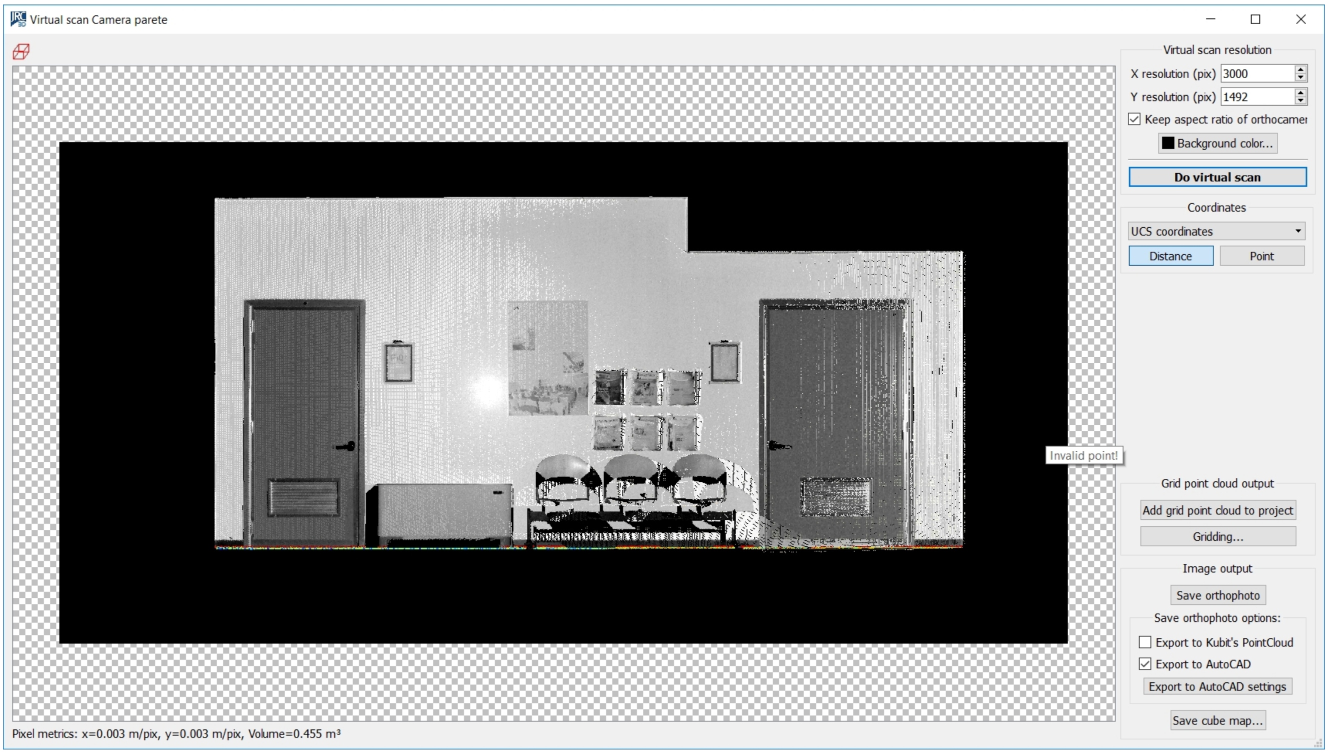
See Cameras for more informations regarding perspective cameras properties and associated functions.

Place here survey point

This tool creates a perspective camera with origin in the current point view. Its properties are the same of the current View Parameters window 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.



It's useful 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). Check 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. The virtual scanner can do also texture blending and mapping for triangulated models.

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: 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

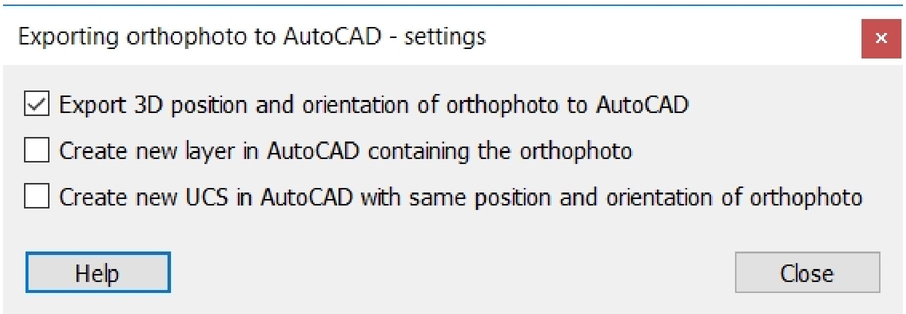
A virtual scan of a plane/ortho camera or projector will be an orthogonal projection and thus an orthophoto. If the image is saved with Save image, a text file is created along with the image file which exports registration information of the ortho image in the scene.

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).
This text file can also be imported in AutoCAD® with the Kubit® plugin.

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.

See also Create Orthophoto.

Exporting orthophotos to AutoCAD



Several options are available to export your orthophoto to AutoCAD. In this dialog, the first checkbox allows you 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 JRC 3D 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.

The second checkbox in the dialog gives choice about 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.

If you selected to export the 3D position and orientation as well, then the third checkbox is enabled. This allows you to choose whether you also want to create in AutoCAD a new UCS with the same position and orientation of the image, or not.

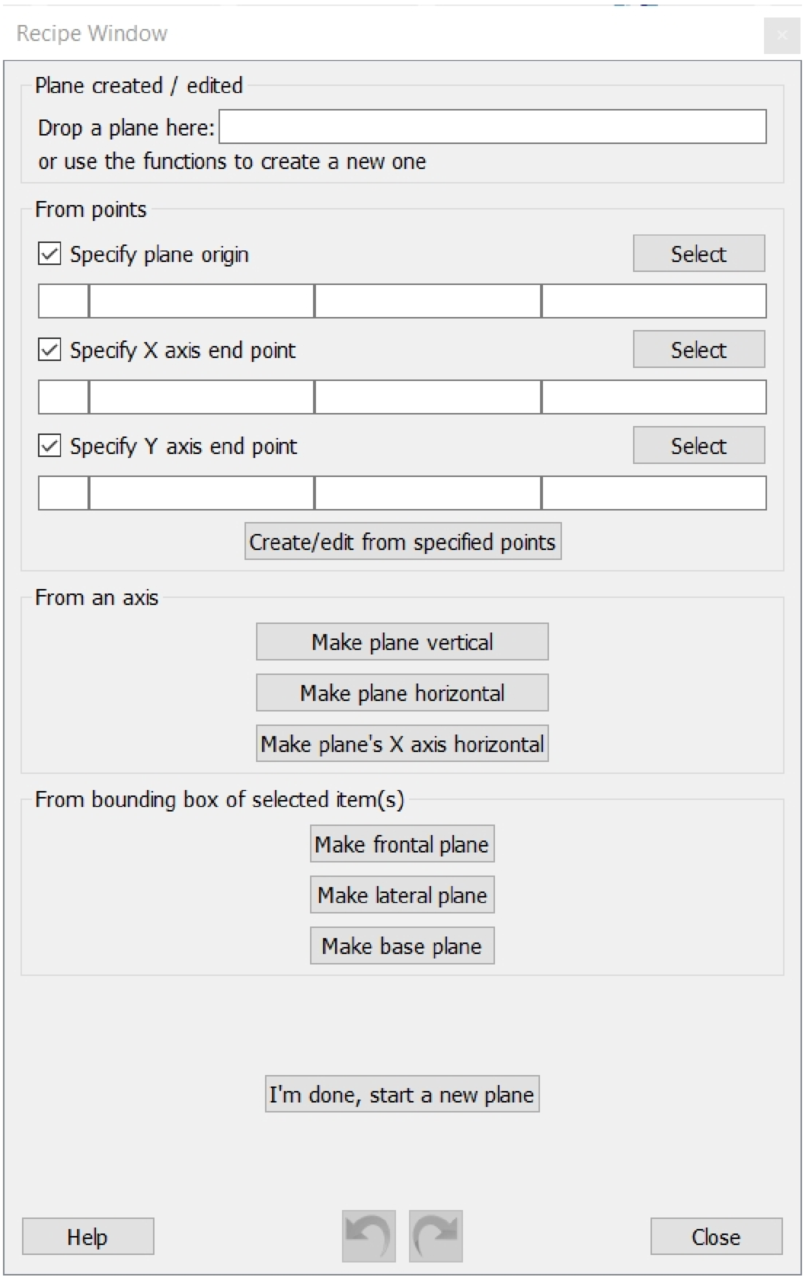
Cross Sections

This submenu contains commands and features to create cross sections and isolines of point clouds and meshes:

- Create/edit plane
- Cross sections
- Cross sections from plane collection

See this videotutorial to learn how to create planes and cross sections from them.

Create/edit plane



This window is a toolbox for creating/ editing your 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.

Creating a plane from points

The superior panel of the dialog allows you to create a plane from, or to make your plane pass by, one to three points. These points are the center of the plane, the X axis endpoint, and the Y axis endpoint. To define one of these points, just click on the corresponding *select* button and Alt + double click on any point in the 3D scene. Otherwise, select any point from the point list window. When you have defined the point(s), click on *create/edit from specified points.* 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 an axis

The central panel contains three buttons whose main function is to adjust the plane's spatial orientation. The first button makes the plane vertical in the current UCS, without moving the central point of the plane. The second button makes the plane horizontal. The third button rotates the plane around its normal vector (passing by its center) until the plane's X axis is parallel to the XY plane of the current UCS.

Creating a plane from model's bounding boxes

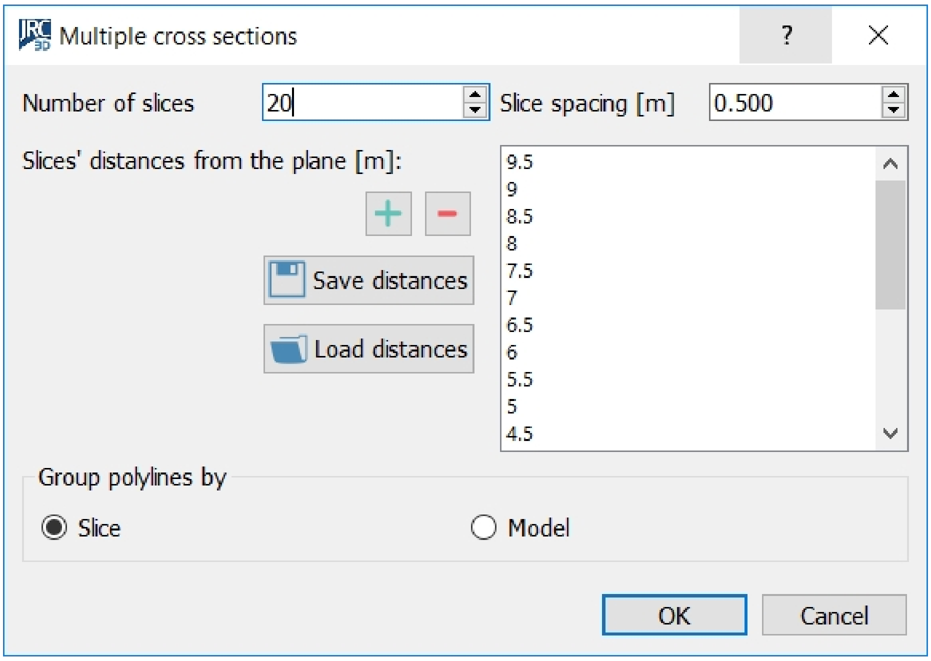
The lowest panel of the dialog allows you to create a plane from the faces of the cumulative bounding box of the currently selected project items. This is useful for example to immediately create the base plane of a certain church or cave, to make plants or sections later.
When you create/edit a plane 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.

Usage tips and tricks

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.*

Cross sections

To create a ***cross sections*** of your models you can access the relative command in the top toolbar or from the planes' contextual 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, one cross section will be created for each model crossed, regardless of how many slices crossed that model and where. 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.

Cross sections from plane collection

This command creates a list of cross section starting from a sequence of planes.

Area & Volume

These tools compute area and volume of point clouds and meshes portions.

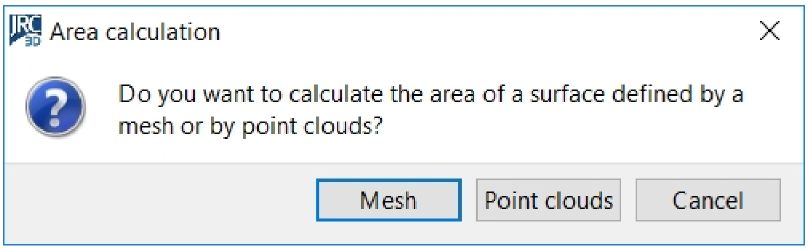
Area

Volume

Compute cut&fill volume

Area

JRC 3D Reconstructor® provides different ways of computing areas.

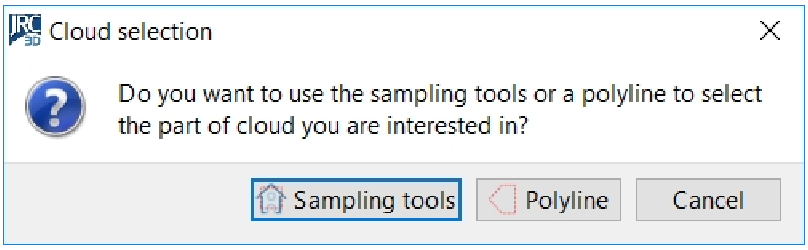


When you press the *Area* button you can choose if you want to compute an area defined by a mesh (1) or by a point cloud (2)

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



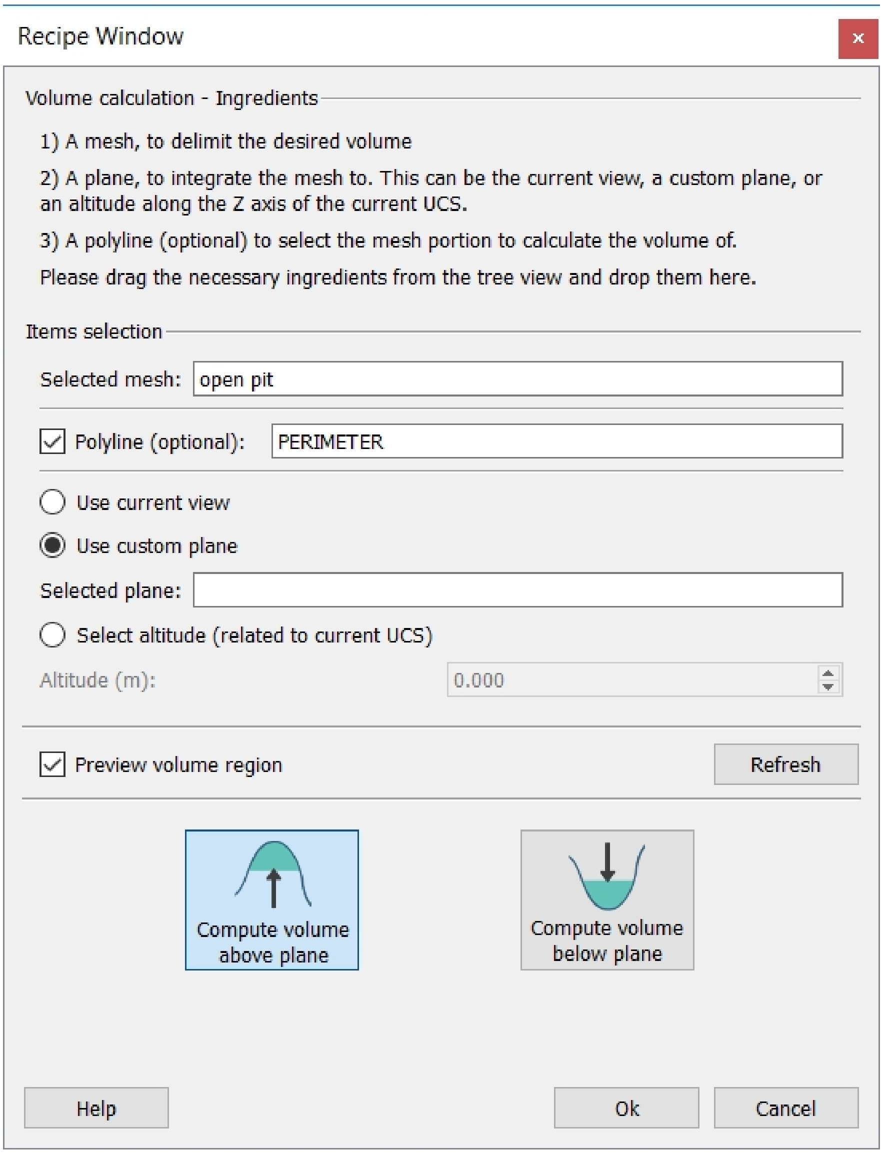
- *selected with Sampling tools*

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



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. JRC 3D 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 JRC 3D 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*. JRC 3D 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.

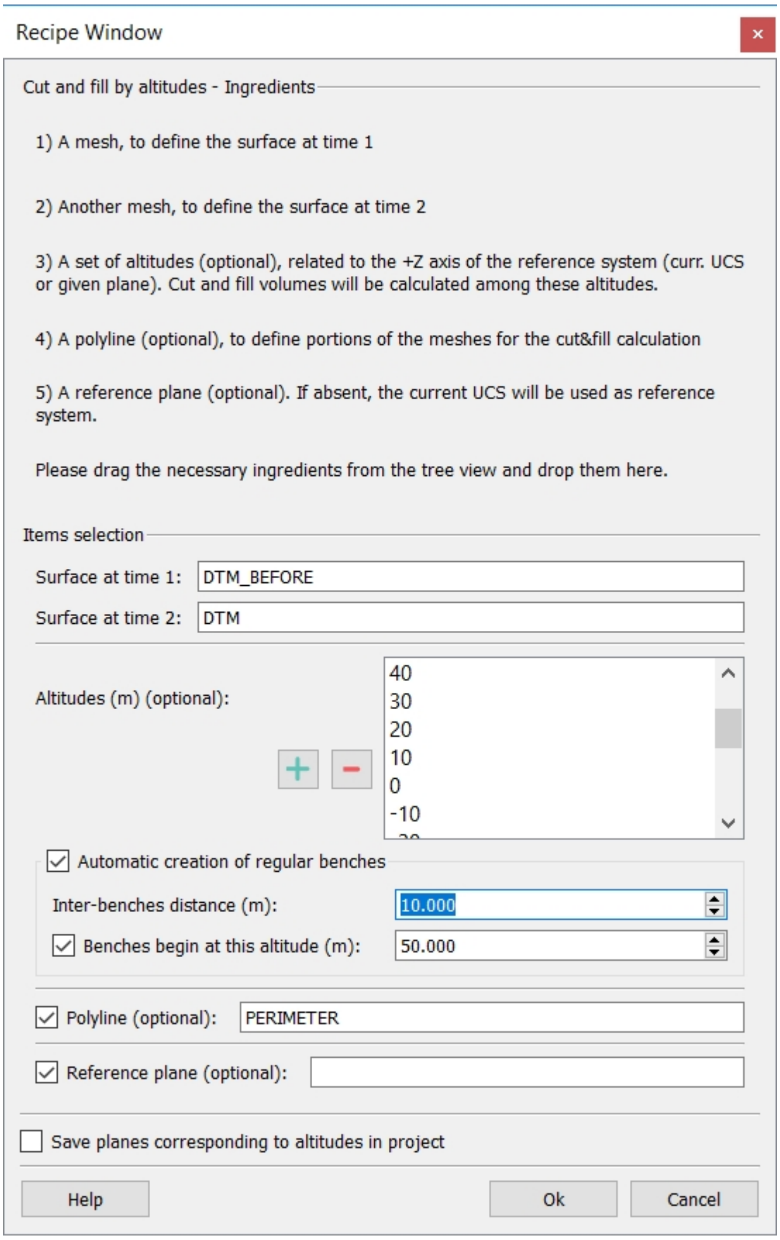
The dialog allows you to fill in a series of fields with the information you want to appear in the report. The fields you uncheck or you left blank will not be included in the report.

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.

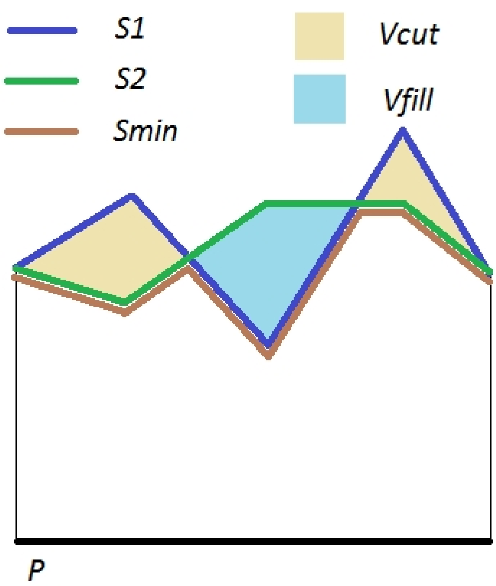
The data you insert are saved also in Windows' registry, therefore the next time you use this dialog you will be able to reuse the data you filled in before.
When you have inserted the data of your report, click on *Save PDF report* and select a filename. The PDF file you specified will be generated.

Compute cut&fill volume



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.

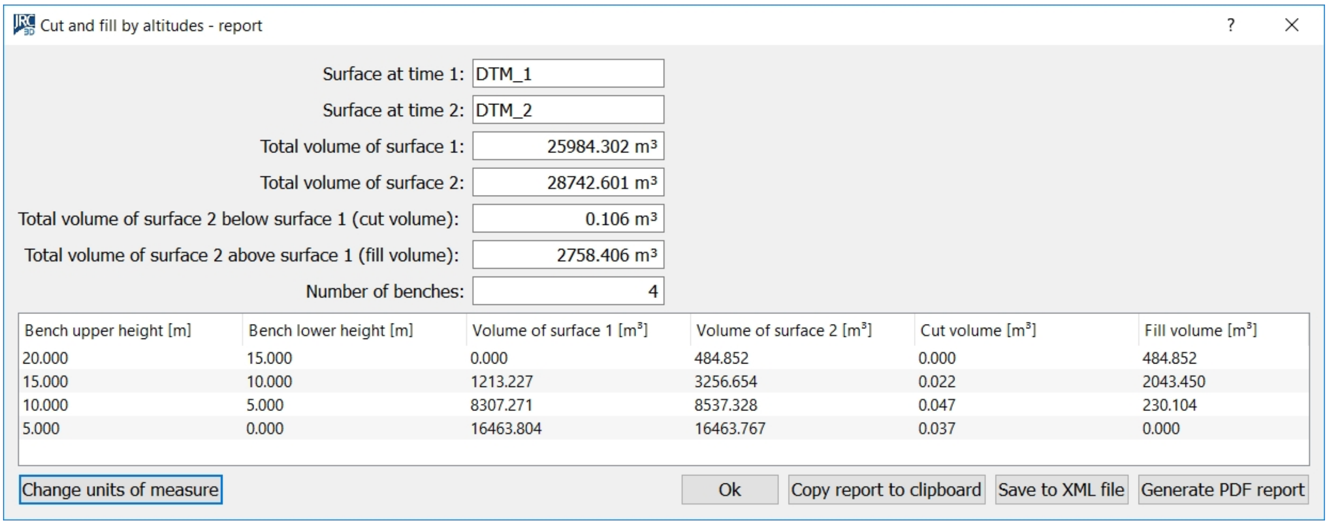
These altitudes are by default referred to the Z axis of the current UCS. However, the user can change this, by dragging and dropping a plane that fills the field *Reference plane (optional)* and becomes the reference of all cut and fill calculation and of all altitudes.

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.

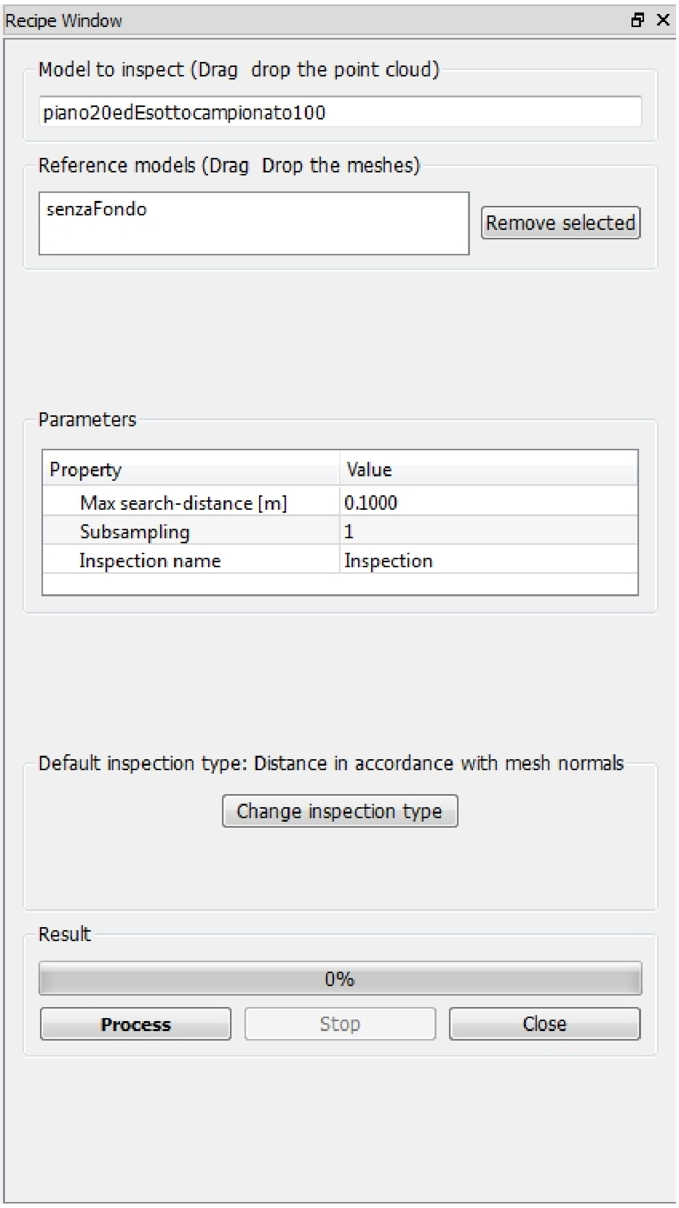
See also Cut&Fill report.

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 the Volume 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.

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 procedure *Inspection* is accessible from *Outputs* → *Inspection* of the top menu bar, from the corresponding button in the top toolbar, or from the contextual menu of any point cloud (structured or not) and any mesh.

The result of the inspection is a new color layer that will be assigned to the point cloud. This color layer carries, for each point its measured distance from the reference model. After performing Inspection 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.

Input models

As inputs, user must provide a model *to inspect* and a *reference* model. The model to inspect must be a *point cloud*. The reference model must be one *triangle mesh*.
User can input the desired models by simply dragging and dropping them, or by pressing *Load model to inspect* and *Load reference model*. Press *Remove* to deselect undesired models from your import.

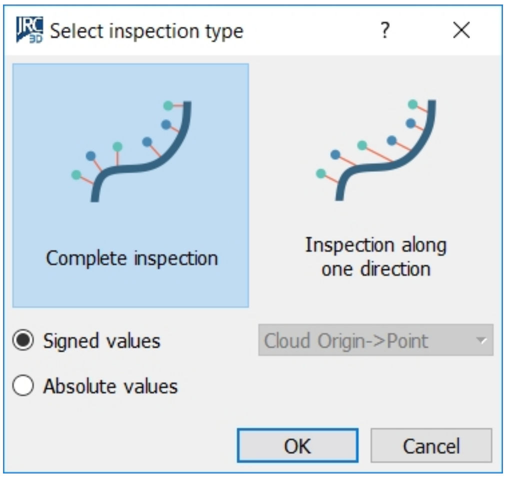
Parameters

At the center of the dialog there is the *Parameters* box that takes:

- **Max 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.
- **Subsampling (factor *s*)**: 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 name**: to define a name for the layer that will be that will be generated

Inspection types

User can select the Inspection type to be performed by clicking " *Change Inspection type*" button.



There are three choices:

- **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*.
- **Signed distances 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).

Finally, press **Process** to start the inspection. The resulting layer is appended to the color type list of the inspected point cloud, with the desired *Inspection name*. Use Colors mapping to optimize the display with a legend.

See this videotutorial to learn how to mage with change detection tool.

Tunnel survey

	**Cylinder virtual scan**	The virtual scanning can here be done in cylindrical projection using a cylindrical projector as a virtual scanner. To properly set the metric pixels refer to the spreadsheet downloaded from the reserved area on gexcel.it site.
	**Generate cross sections**	To generate a multiple cross section (with a defined spacing between planes); all of them are normal to a selected flythrough, a trajectory that describes the elongation of a tunnel.

See also this videotutorial that shows the whole process.

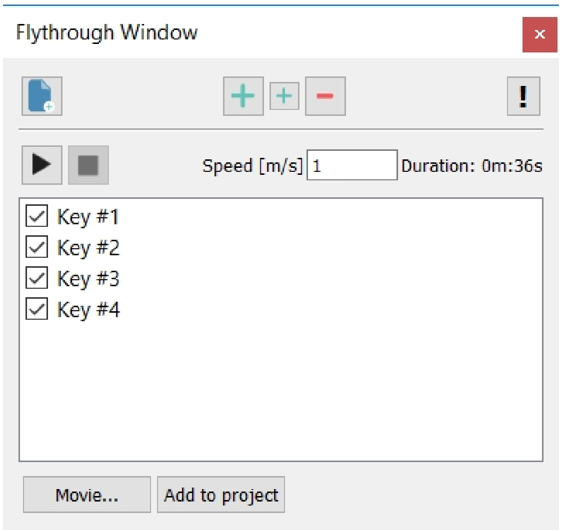
Video Record

Video Record

In JRC 3D Reconstructor® the creation of a video of your 3D model is possible, by using the *Outputs->Video Record* commands:

- **Flythrough Editor** (shortcut key Ctrl+F): to define and edit a trajectory that the “virtual eye” of the 3D world can go through
- **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 Window



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

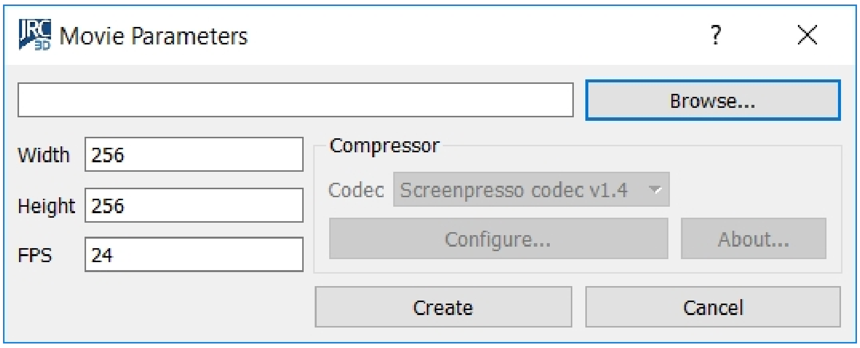
- **Add key**: appends the current view point to the list
- **Insert key before**: inserts the current view point before the selected key in the list
- **Delete key**: deletes the selected key
- **Clear**: clears the list
- **Update trajectory**: if at least 2 keys are defined, a temporary trajectory 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 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.

Movie Dialog

Movie Dialog



This dialog allows you to create a movie out of a video trajectory in your 3D scene.
Input the video file name, the movie resolution, the frames per second (FPS) and press Create. Then, a dialog appears and asks to select the desired video codec (between those present in your PC) and compression parameters. Subsequently, video encoding starts.

See also flythroughs, and flythrough editor.

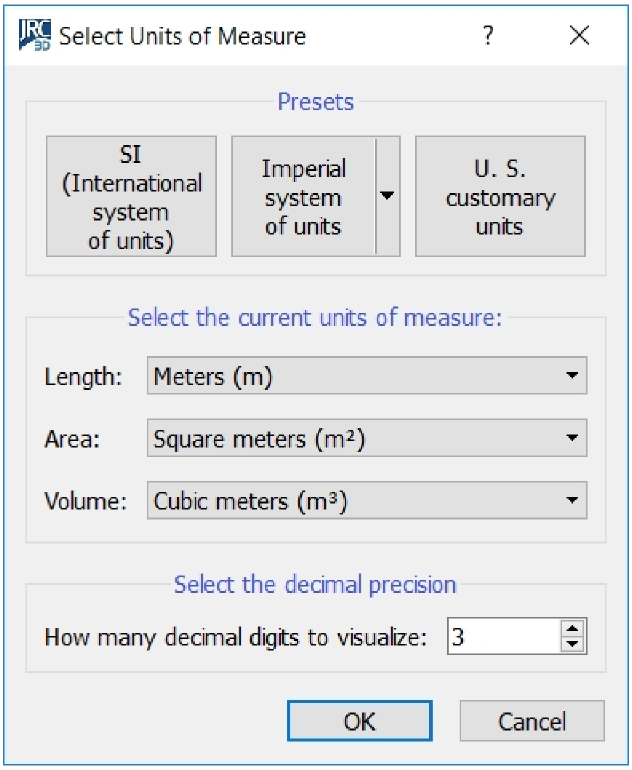
Drawing tools

	Constrain draw to plane	To turn on the function to pick 3D points in the scene in a way that their projections on a formerly chosen plane are selected and shown in the point list window.
	Link to AutoCAD	Before activating this function, AutoCAD with the Kubit plugin must be running. When the link is enabled, all the points selected in *Point Pick mode* or in the Virtual Scan dialog or in the Edit 2D grid dialog are sent to AutoCAD.

Save snapshot

	Save snapshot	This function allows you to capture a snapshot of the 3D scene with its rendered models. You can save the captured snapshot in a variety of image formats, including Bitmap, Jpeg and PNGSave snapshot.

Manage units of measures



This dialog allows you to set the preferred units of measure for lengths, areas and volumes; and to set how many decimal digits should appear in JRC 3D Reconstructor® 's output data. This function, therefore, gives you a great amount of countrol on how lengths, areas and volumes should be formatted in all Reconstructor windows, outputs, PDF reports, etc.

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 JRC 3D 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 JRC 3D 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

Tools menu

Here the In JRC 3D Reconstructor® several processing tools are achievable.
You can find most used tools also in the Top Toolbar.

- **Go To LineUp**
  - File
  - Navigation
  - Tools
- **Pose & Registration**
- **Selection & Fitting Tools**
- **Points Filtering & Clustering**
- **Meshing**
- **Photo & Color**
- **UAV & Imaging**

LineUp®

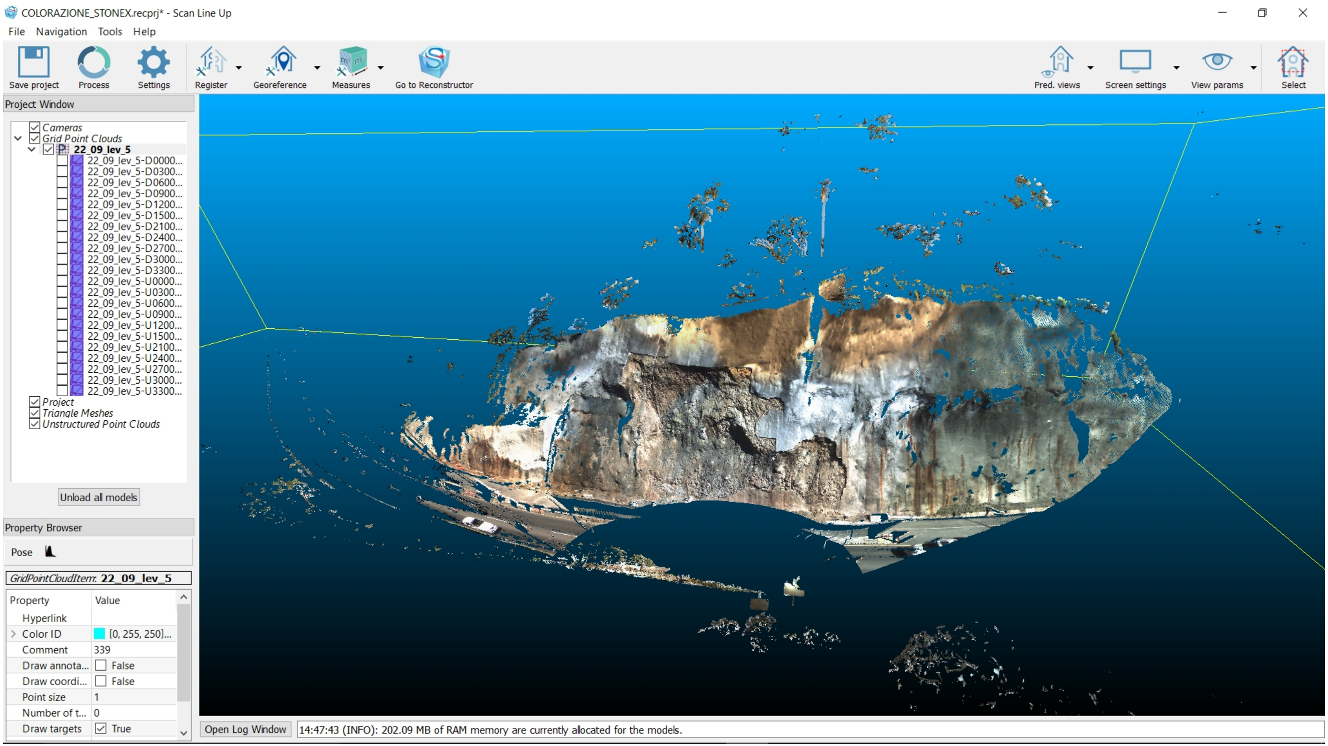
LineUp® is a tool in JRC 3D Reconstructor® that easily allow you to:

- Import
- Preprocess
- Register
- Georeference

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

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

By clicking on *Go to LineUp*, starting from JRC 3D Reconstructor®, you can go in LineUp® environment.



The LineUp® menu is subdivide in the following command groups:

- File
- Navigation
- Tools

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

Save Project
To save the project

Process
To open the Scan Processing Wizard and import, preprocess, preregister and register any set of scans

Settings
To change all the setting parameters of the different steps of the workflow (see “More Settings” in the *Scan Processing Wizard)*

Register
To register the imported scans. This command includes both pre-registration process (target-free or target-based) and fine registration

Georeference
To georeference the models using known control points

Measures
To take measures on the point clouds

Go to Reconstructor
To return in JRC 3D Reconstructor® environment

Navigation commands
To navigate in the LineUp Window

Selection tools
To select a region of point cloud and delete the area inside or outside it

LineUp® Pro
LineUp® Pro is a *LineUp®* extension that includes the same features of LineUp® tool to which is added the automatic target-less registration.

File in LineUp®

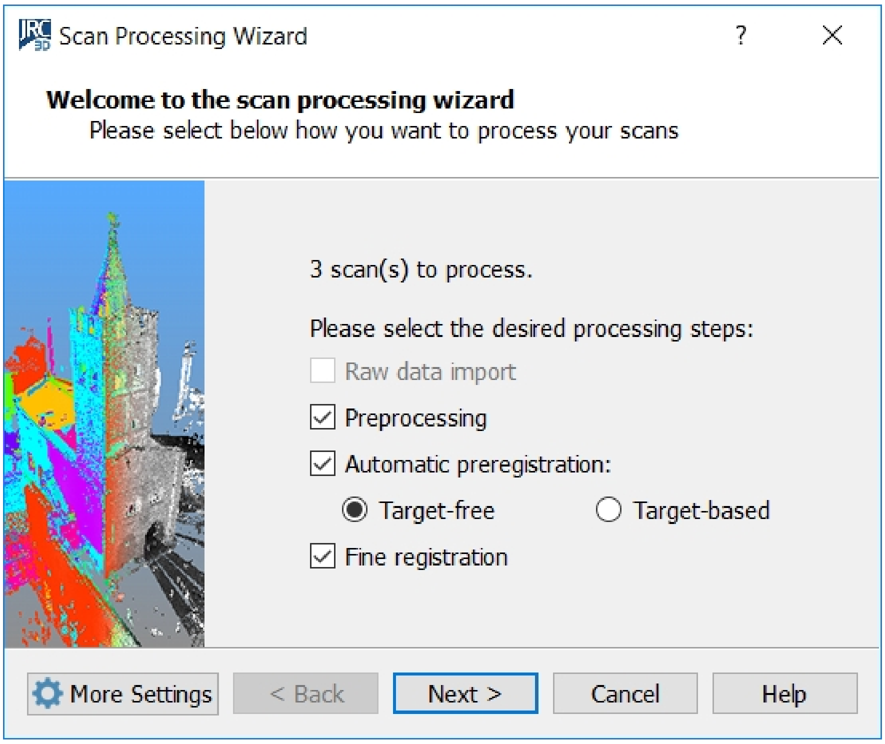
Save Project
This function saves the current project, safley storing all the project items, their load/unload state, their properties, the current 3D camera pose, etc.
It is advised to save the project frequently, especially before long processing operations.

Import point clouds
To import every point clouds format supported by JRC 3D Reconstructor®

Import sketch from LineUp® Notes

Go to Reconstructor
To return in JRC 3D Reconstructor® environment

Scans Processing Wizard

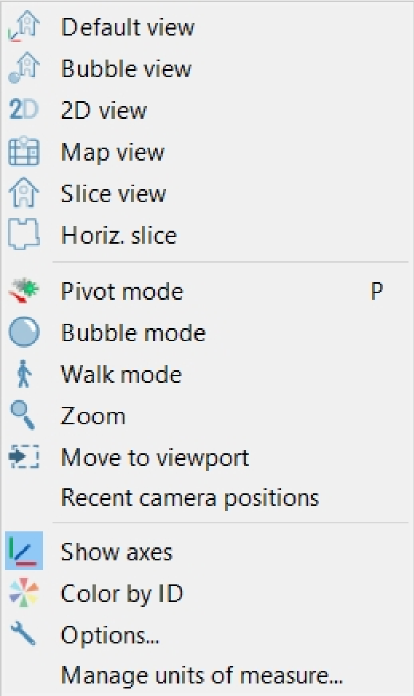


This wizard allows the user to

- Import
- Preprocess
- Preregister
- Register

any set of scans in LineUp® tool.

Navigation in LineUp®



Navigation in LineUp® tool is very similar to that of JRC 3D Reconstructor®:

- *Predefined views*
- *Navigation modes*
- *Screen settings*

In addition, other predefined views are available:

- Slice View
- Horizontal Section View

These modes can be used to check the correct alignment along a vertical/horizontal section of the displayed items.

Also Manage units of measure is here present.

Tools in LineUp®

LineUp® features the following categories of tools.

Point Filtering & Clustering
To filter/edit point clouds and to cluster them

Pose & Registration
To register and georeference point the point clouds and 3D models

Select
To select a region of point cloud and delete the area inside or outside it

Measures & notes
To take measures of distance and screenshot of the 3D window

Settings
To change all the setting parameters of the different steps of the workflow (see “More Settings” in the Scan Processing Wizard)

Process
To open the Scan Processing Wizard and import, preprocess, preregister and register any set of scans

See also at the relative voices in the general Tools command in JRC 3D Reconstructor®.

Points filtering and clustering in LineUp®

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 cloud, some of them work only on grid point clouds.
In LineUp® the following filtering and clustering tools are present:

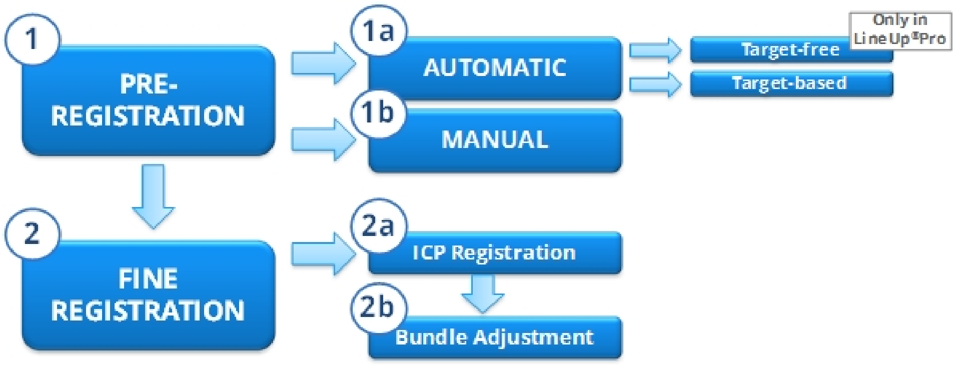
- Pre-process Grids

- Restore raw data
- Restore deleted points
- Level 3D density of clouds
- Make single cloud
- Remove duplicate points

Pose & Registration in LineUp®

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.

Registration

- Automatic, target-free preregistration
- Automatic, target-based registration
- Manual pre-registration
- Manual pre-registration among grid point clouds
- ICP cloud-to-cloud registration
- Global fine registration (Bundle Adjustment)
- Scan alignment per groups

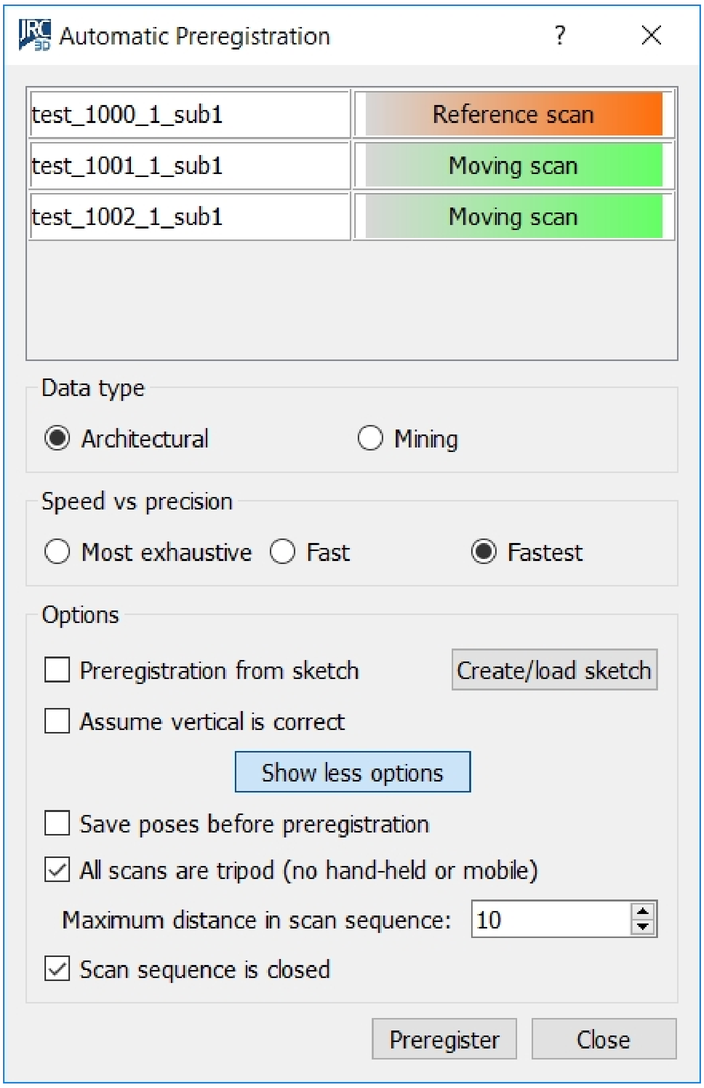
Georeferencing

- Point cloud georeferencing
- Pick reference point

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.

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 JRC 3D 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

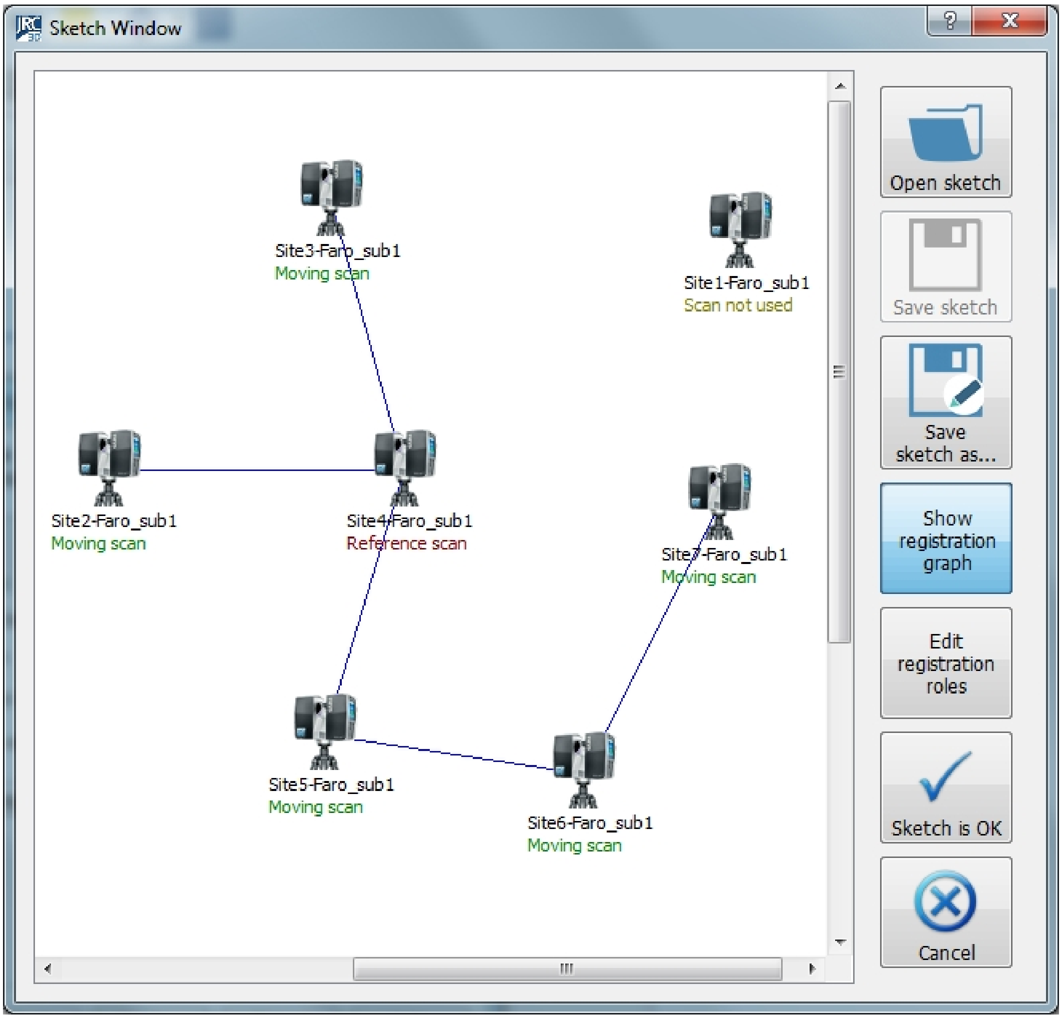
scans are already on their vertical (with a confidence of 2-3°), you can tell the algorithm to leave the scans' vertical unchanged. If you don't, LineUp Pro will also esteem the scans' vertical, with the guarantee to correct angles between the given vertical and the correct one of max. 30°. As additional 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, JRC 3D 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, JRC 3D Reconstructor® will try to match scan 5 with all the following, until scan 15. Scan 5 and scan 16 will not be matched because JRC 3D 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 JRC 3D 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 JRC 3D Reconstructor® or in LineUp Notes. 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. If the sketch is loaded from a project created with Line Up Notes, it may show also a site orthographic plan as background. 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:

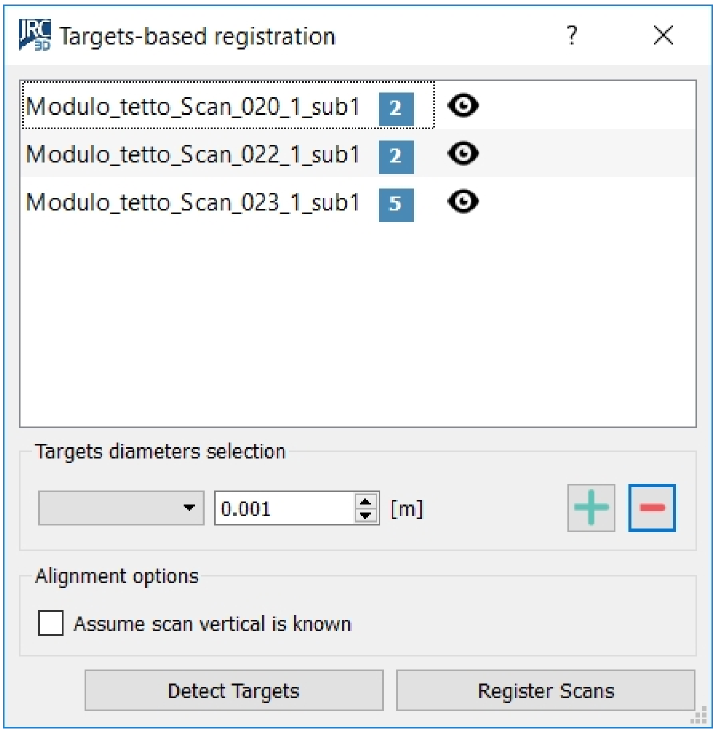
- **Reference scan**. The scan is not moved during the automatic preregistration.
- **Moving scan**. The scan will be preregistered against the others during the preregistration.
- **Scan not used.** The scan will not be preregistered. This is useful if you acquired scans that are not useful for the 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, otherwise you can open a sketch that you have imported from the iPad app "LineUp Notes".
- **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.

Automatic Target-based 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 user can see the number of detected targets. 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

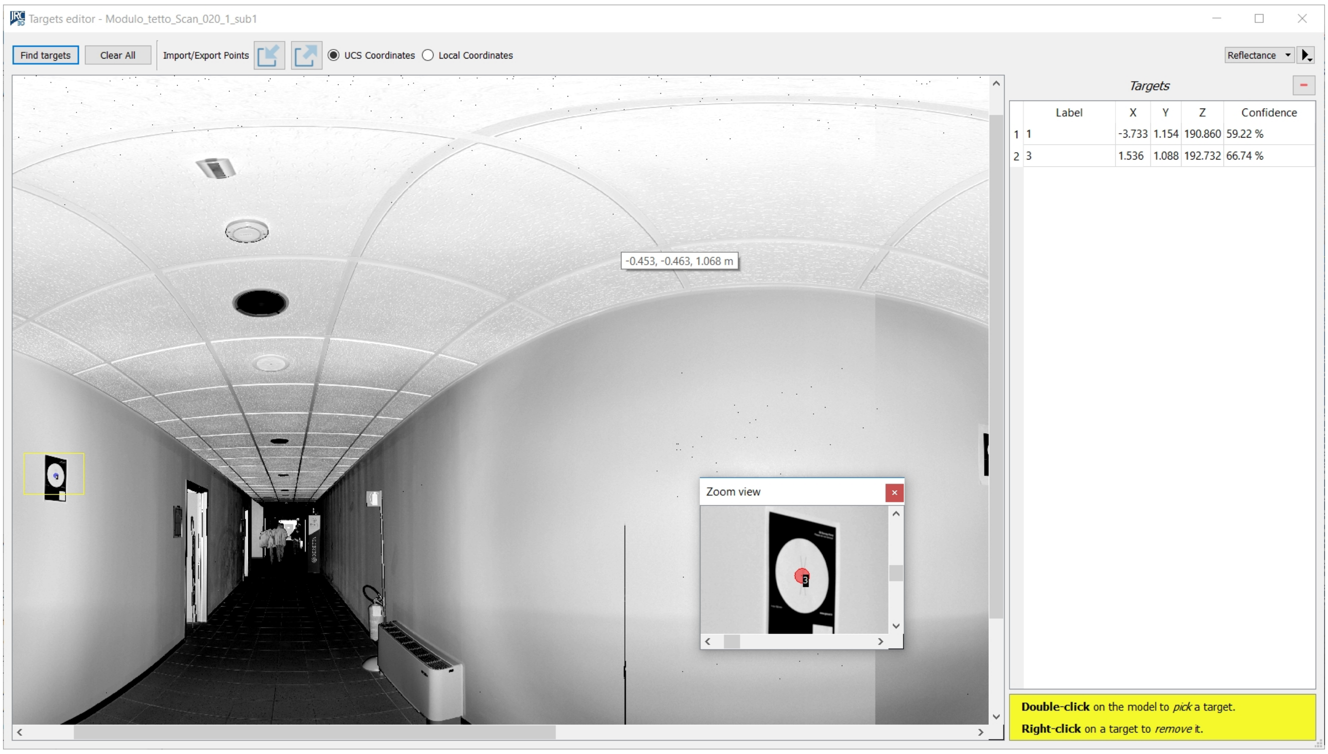
targets associated with the selected scan are overwritten.
The *Register Scans* button performs both the targets detection and scan registration stages. The detection phase is skipped for scans with already associated targets.

Options

Targets diameters selection: 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.
Alignment options: The user can select if the scans vertical is to be assumed known.

Note: it's necessary before to compute the confidence of the point clouds (see Pre-processing Grids) to help the process algorithm to find the target. At least 5 point cloud's points (on the same polyline) are needed to fit the contour of the target (an ellipse).

Target 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 targets* button.

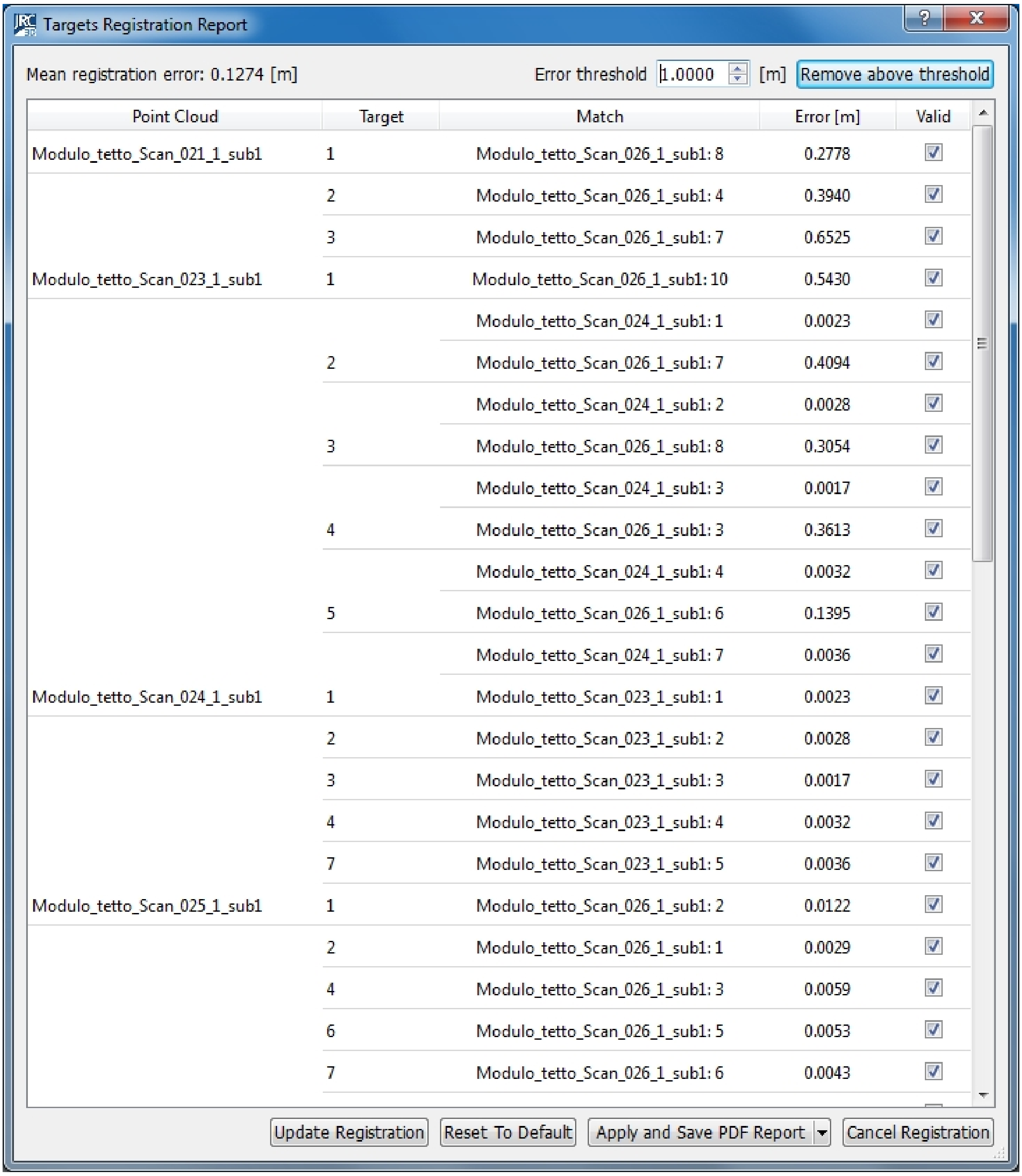
The mouse controls are the following:

- double click to add a 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 are listed the current targets with their 3D coordinates and confidence values. 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.

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

Targets Registration Report



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.

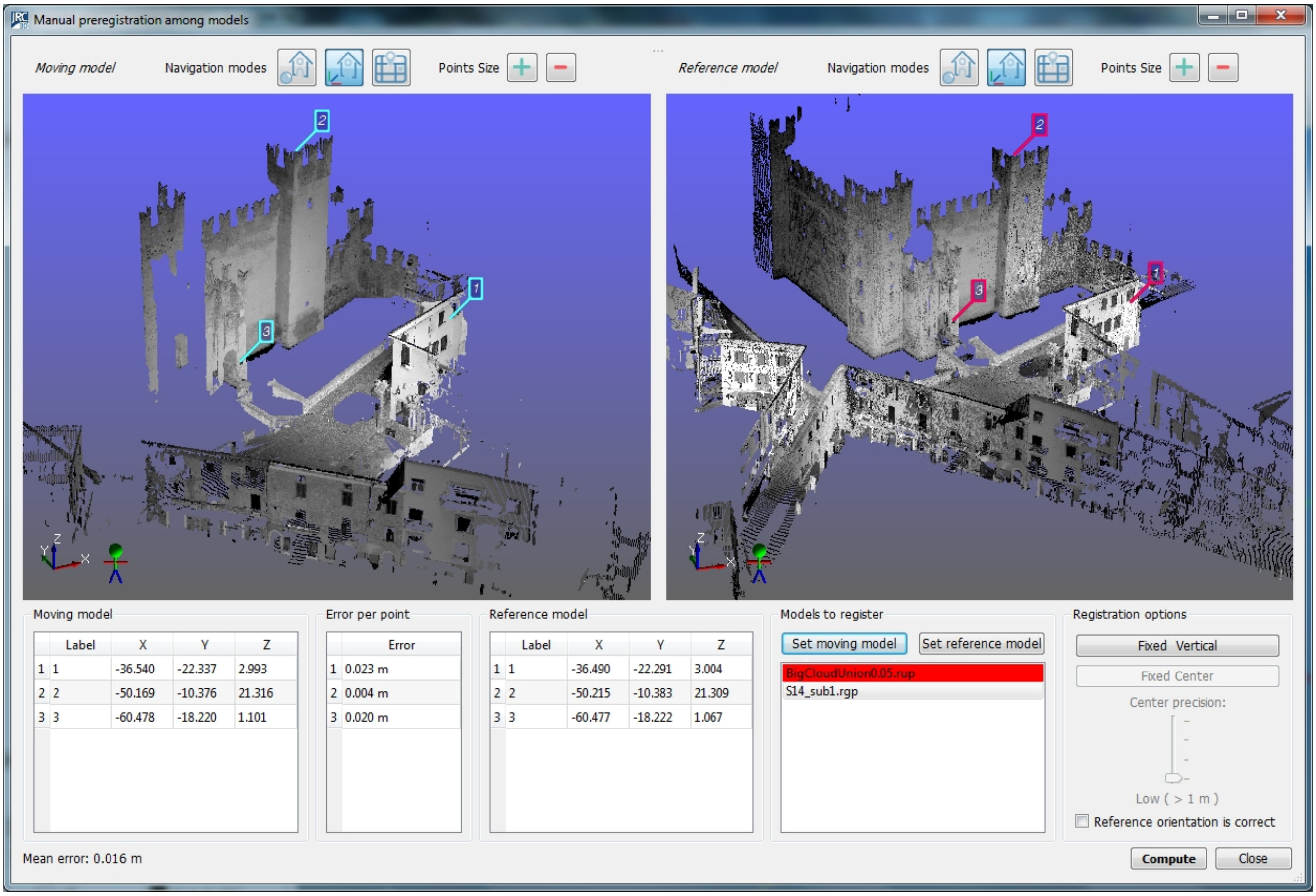
You can uncheck a matching target pair, by pressing the checkbox on the right, and press *Update registration* on the bottom left. The registration will be recomputed without the outlier and the mean registration error will improve accordingly.

Please note that the alignement 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 uncheck 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.

Manual Pre-registration (among models)



Manual preregistration quick guide

1. Select at least two models from JRC 3D 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 JRC 3D 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.

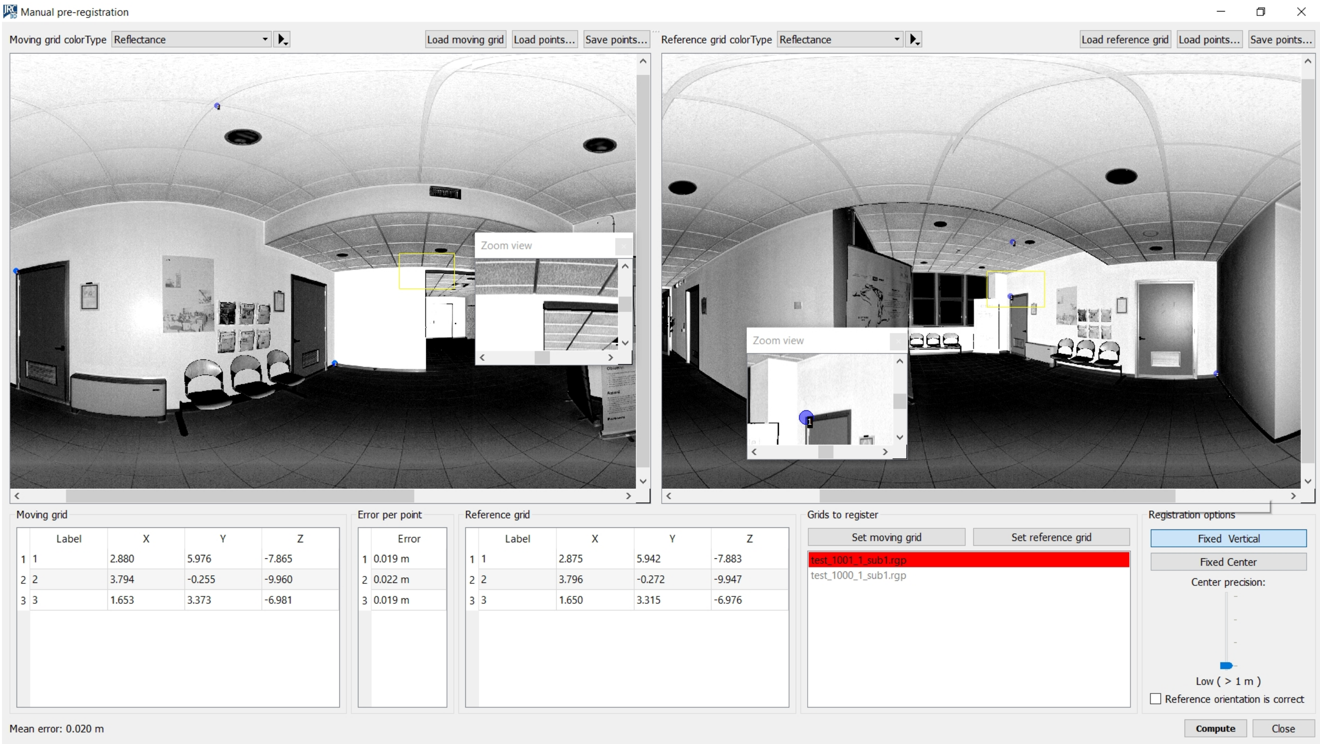
Registration constraints

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 JRC 3D 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).

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 JRC 3D 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

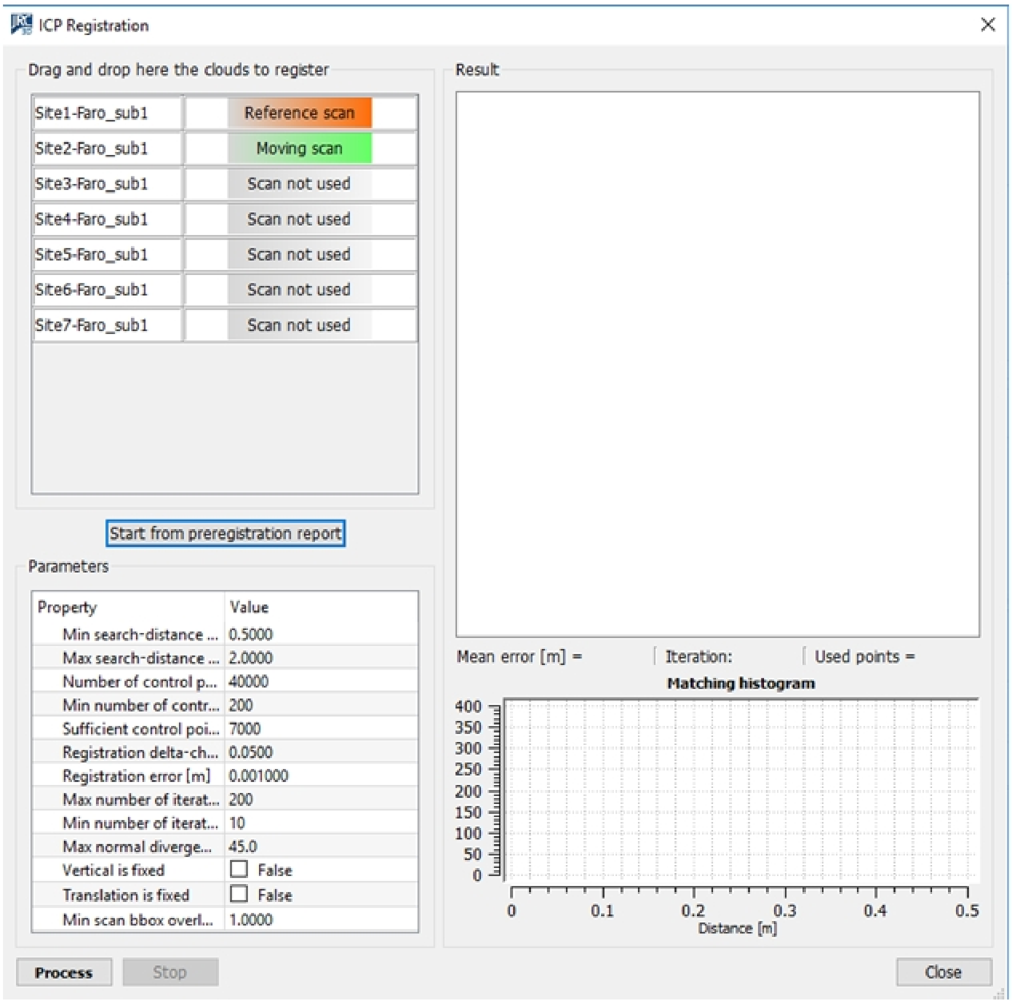
Space bar	Rotates the range image of 90°
Shift + LMB + move mouse	Translates the range image
RMB on a selected point	Deletes the point
LMB on a point + move mouse	Moves the point around
Alt + central mouse button	Fit image to window
Alt + RMB	Reset image to original size

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 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.

ICP Cloud to Cloud 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.

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. JRC 3D 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.

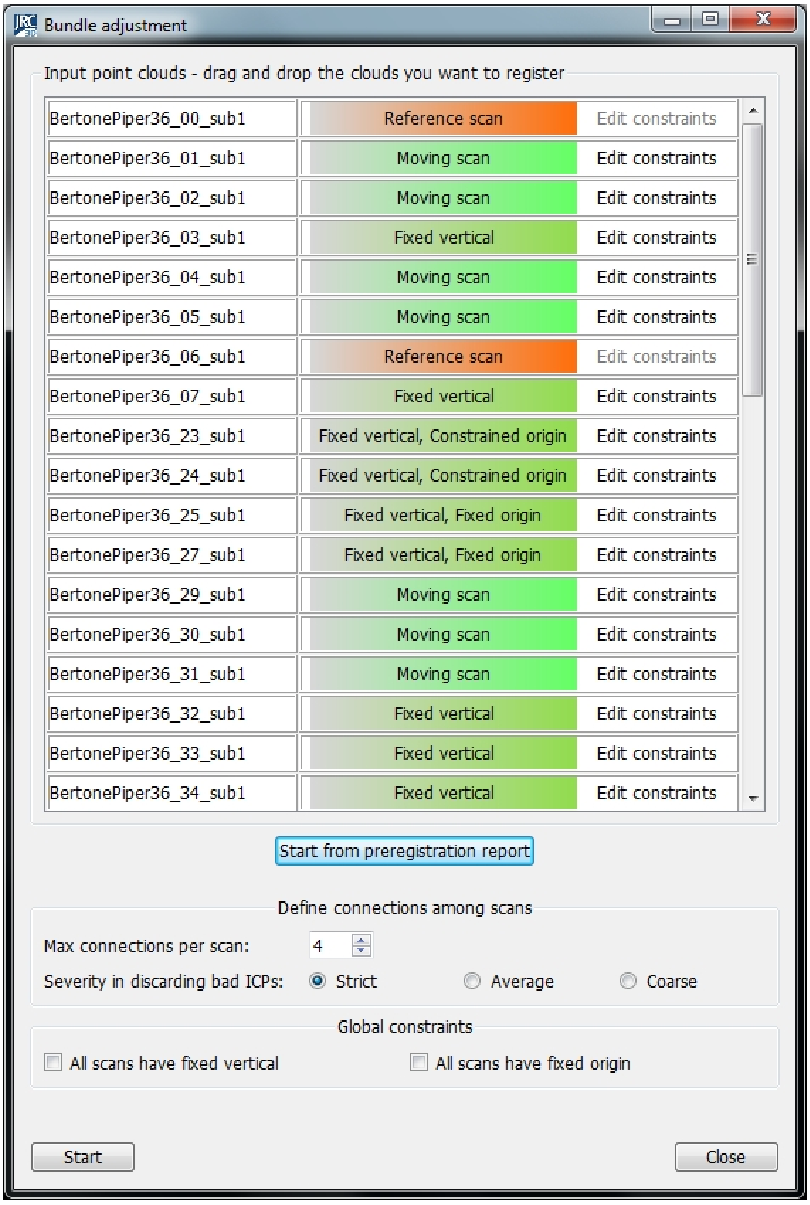
Finishing the registration

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.

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 (unconstrained), 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.

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.

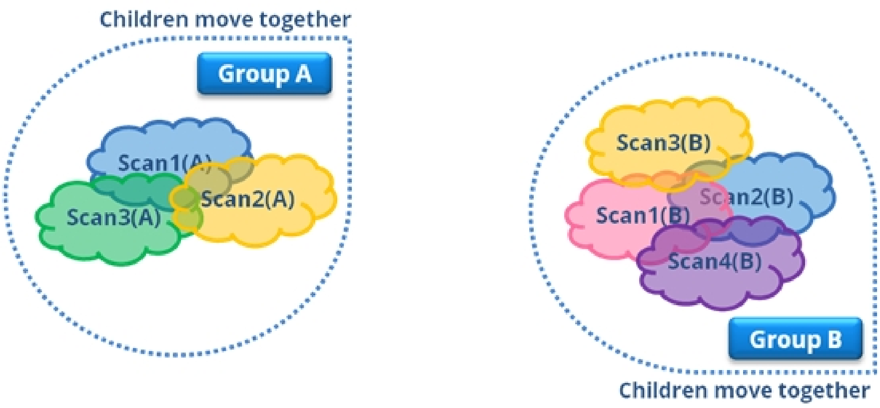
A Registration role for all the scans inside the group can be set: *None*, *Same as parent*, *Children move together.* These roles are used to carry out a registration through groups.

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

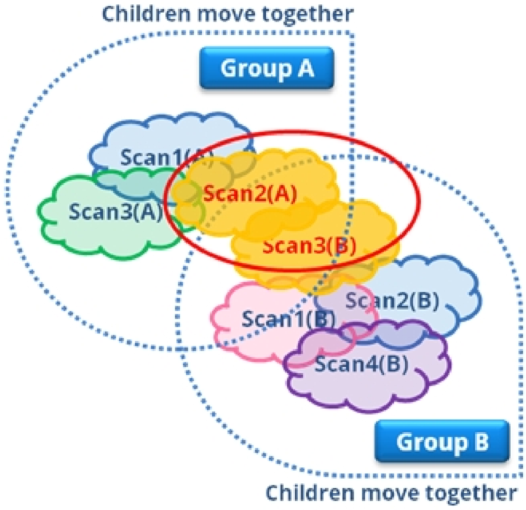
1. Find the groups of scans already registered.
   There are two possibilities to create groups:
   a. Automatically
      When you automatically register scans with the LineUp® tool, groups containing just registered scans are created by default
   b. Manually
      you can manually create a group already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



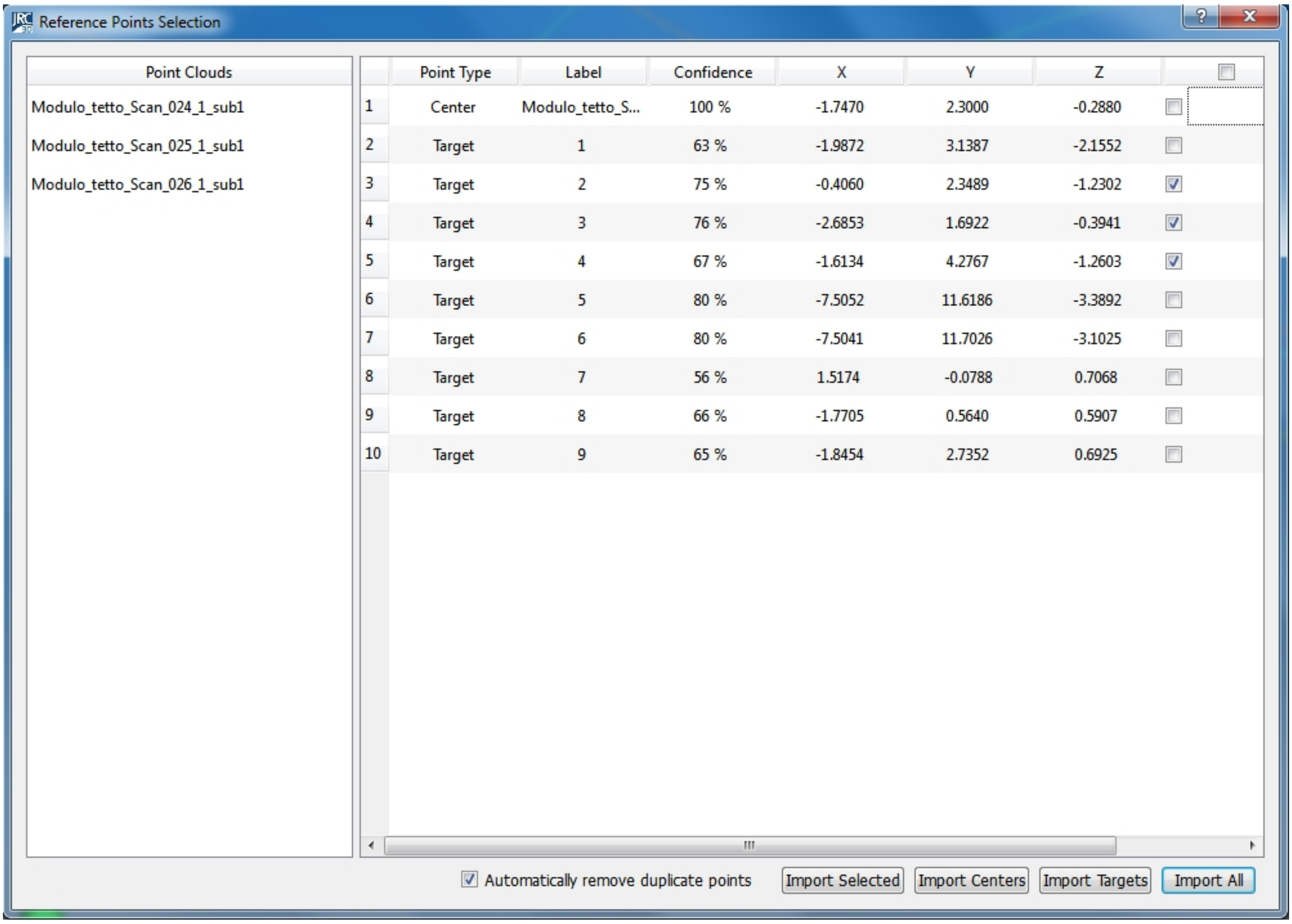
Since the "*Children move together*" registration role of the groups permit a rigid and common movement of all the scans of the groups, this procedure is enough to align all the scans.

Point cloud 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

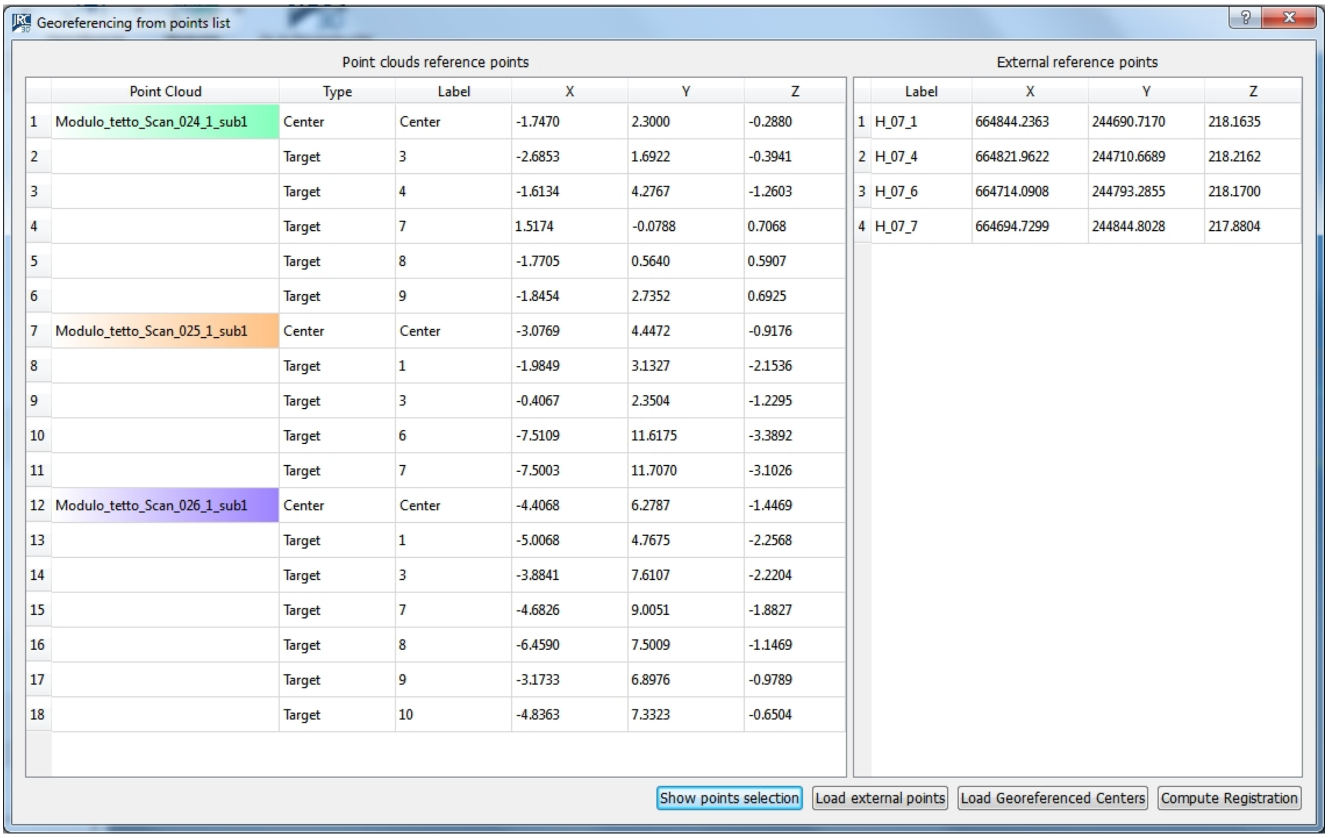


This dialog allows the selection of multiple reference points from the point clouds target and center points.
Target points can be automatically or manually set using the targets registration dialog or targets editor dialog tools.

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



Georeferencing from points list

Point clouds reference points

	Point Cloud	Type	Label	X	Y	Z
1	Modulo_tetto_Scan_024_1_sub1	Center	Center	-1.7470	2.3000	-0.2880
2		Target	3	-2.6853	1.6922	-0.3941
3		Target	4	-1.6134	4.2767	-1.2603
4		Target	7	1.5174	-0.0788	0.7068
5		Target	8	-1.7705	0.5640	0.5907
6		Target	9	-1.8454	2.7352	0.6925
7	Modulo_tetto_Scan_025_1_sub1	Center	Center	-3.0769	4.4472	-0.9176
8		Target	1	-1.9849	3.1327	-2.1536
9		Target	3	-0.4067	2.3504	-1.2295
10		Target	6	-7.5109	11.6175	-3.3892
11		Target	7	-7.5003	11.7070	-3.1026
12	Modulo_tetto_Scan_026_1_sub1	Center	Center	-4.4068	6.2787	-1.4469
13		Target	1	-5.0068	4.7675	-2.2568
14		Target	3	-3.8841	7.6107	-2.2204
15		Target	7	-4.6826	9.0051	-1.8827
16		Target	8	-6.4590	7.5009	-1.1469
17		Target	9	-3.1733	6.8976	-0.9789
18		Target	10	-4.8363	7.3323	-0.6504

External reference points

	Label	X	Y	Z
1	H_07_1	664844.2363	244690.7170	218.1635
2	H_07_4	664821.9622	244710.6689	218.2162
3	H_07_6	664714.0908	244793.2855	218.1700
4	H_07_7	664694.7299	244844.8028	217.8804

Show points selection | Load external points | Load Georeferenced Centers | Compute Registration

This dialog allows the user to load external reference points for geo-referencing scans using previous selected target points.

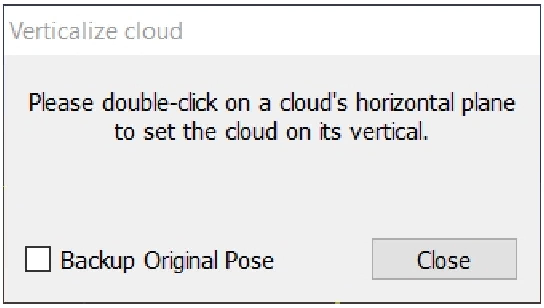
The buttons in the bottom part of the dialog have the following functions:

- **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. A registration report is shown at the end of the process.

Pick Reference point

This function opens the Targets editor dialog.

Determine the cloud's vertical direction



Determine the cloud's vertical direction command is available in the *Pose&Registration* tools.

It's used to define a plane that should be horizontal (e.g. a floor) by double clicking on it and to set the vertical direction of the point cloud (z axis) as the normal direction of this plane, by rotating the point cloud.

Select in LineUp®

You can find the command *Select* in *Tools* or in the upper right side of the Top tool bar in LineUp® environment.
You can switch between navigation and selection mode by pressing the spacebar.

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

Rectangle
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
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
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*

There's two ways to delete points just selected:

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

Delete outside
The points that fall outside the selected region are deleted, and can be undeleted at a later moment.

See also Restore Deleted Points to learn how to undelete points.

Measures & notes

Measures & notes in LineUp®

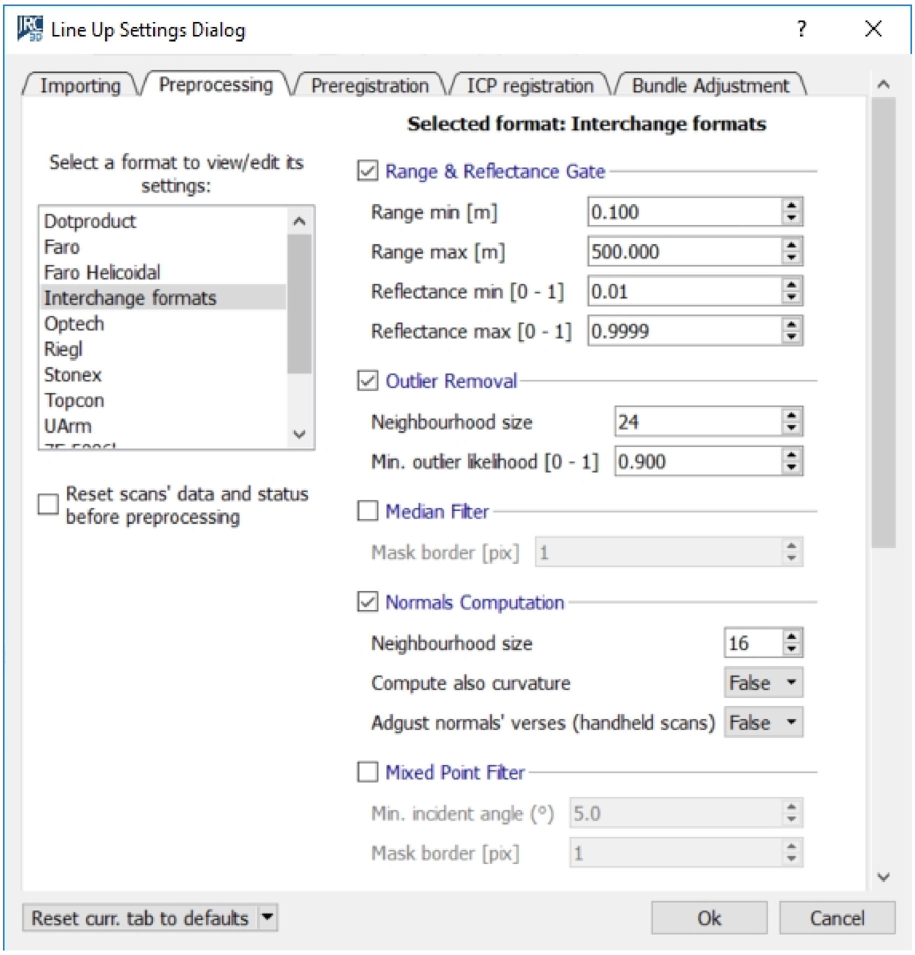
In LineUp® tool you can take:

Distance This button allows you to measure a distance between any two points in the 3D scene. After activating this button, press the Left Mouse Button of the mouse (LMB) for the first point and keep it pressed while dragging the mouse to the second point, then release the LMB. A result dialog will pop up displaying the distance between the two points. You will be offered the option to save the distance as an annotation in your project.

Save snapshot

This function allows you to capture a snapshot of the 3D scene with its rendered models. You can save the captured snapshot in a variety of image formats, including Bitmap, Jpeg and PNGSave snapshot.

LineUp® Settings Dialog



In this dialog, the user can specify advanced settings for Line Up®'s scans processing workflow.

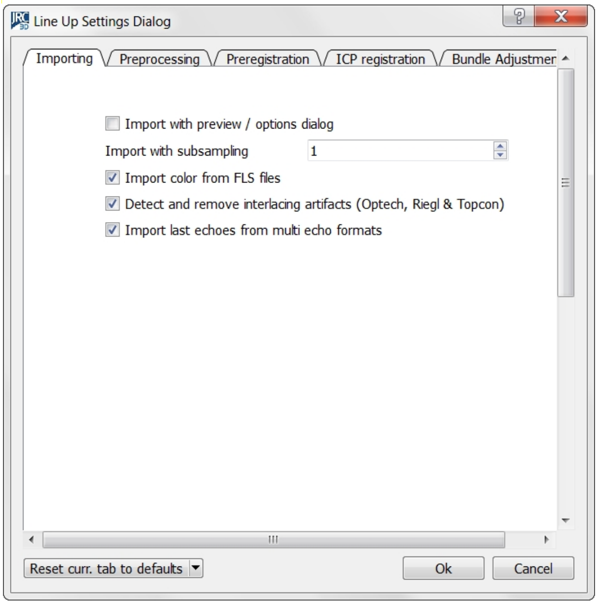
See also Point Filtering & Clustering and Pose & Registration.

Process (LineUp®)

This function opens the Scan Processing Wizard.

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

LineUp Setting Dialog



The *LineUp Setting Dialog* is available in the Process command.

It's splitted in several tabs, concerning:

- Importing
- Pre-processing
- Pre-registration
- ICP registration
- Bundle Adjustment

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

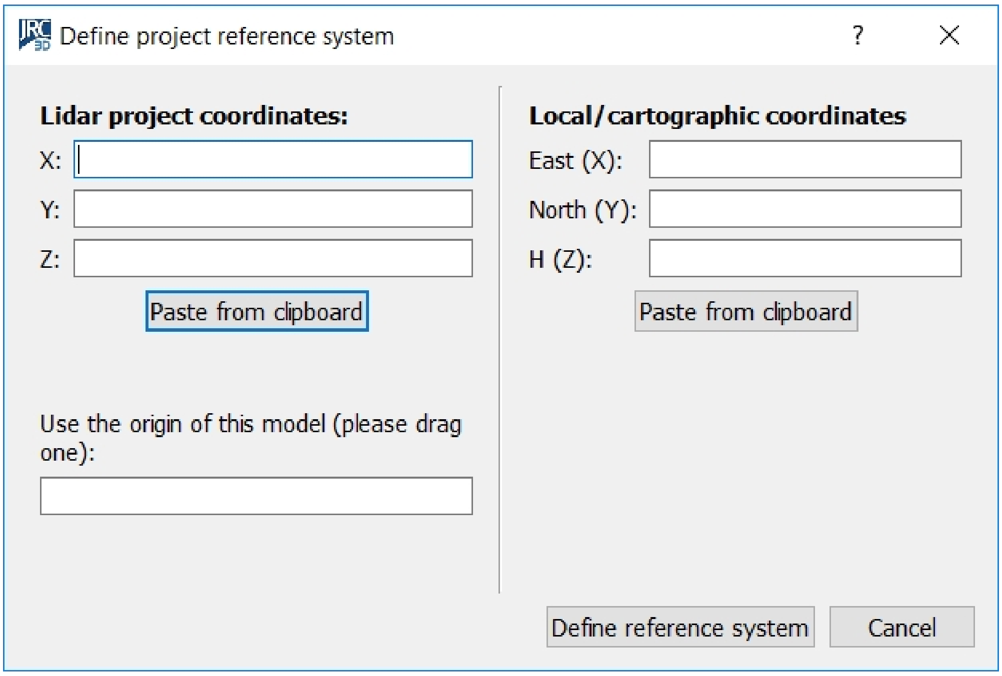
See details in Import, Point Filtering & Clustering and Pose & Registration.
See also Getting Started to learn how to follow an efficient workflow for your projects.

Pose & Registration

JRC 3D Reconstructor® includes various registration techniques:

- Define Project Reference System
- Manual Positioning
- Point List Registration
- Pose Transform

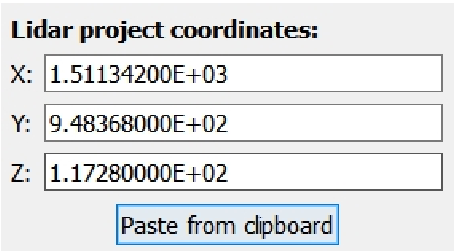
Define project reference system



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.

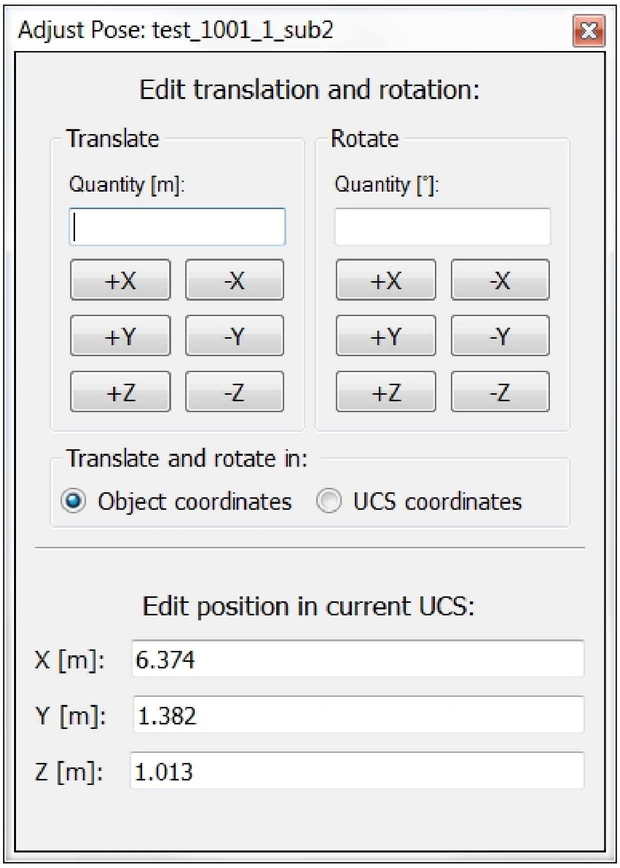
You can simply copy the coordinates of a point (as below written) and paste them with “Paste from clipboard” button filling the empty spaces.

P25 *1511.342* *948.368* *117.280*



For more sophisticated methods of georeferentiation, see also Georeferentiation.

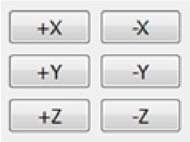
Manual positioning (Adjust Pose)



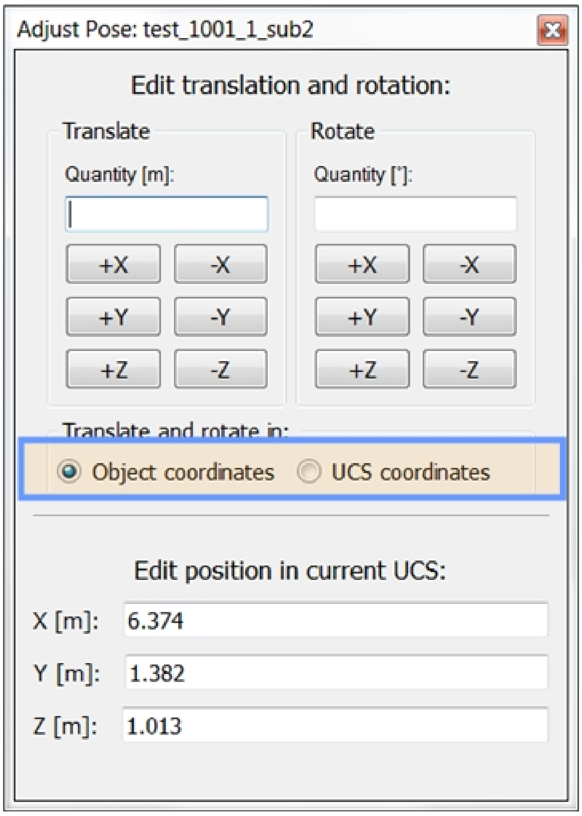
This window is enabled only if an item in the Project Window is selected. Then also the bounding box of the current item is highlighted in bold yellow.

It' possible to manually change the pose of an object by using the *Manual Positioning window/Adjust Pose* (press CRTL+A).

Insert the desired quantity of translation (meters) and rotation (degrees) and press the positive and negative buttons



to apply transformation.

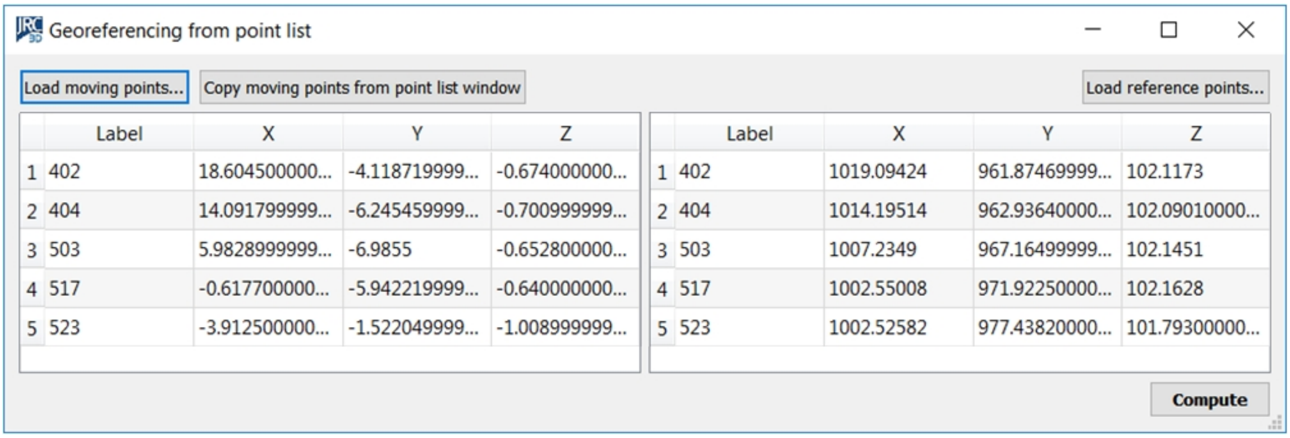


If **Object coordinates** is enabled, the rotation is around the origin of the object's coordinate system and the translation is along the axes of the same system.

If **UCS coordinates** is enabled, the rotation is around the current UCS' origin, and the translation is along the axes of the same UCS.
A direct changing of the 3D position

See also the Pose dialog.

Georeferencing from point list

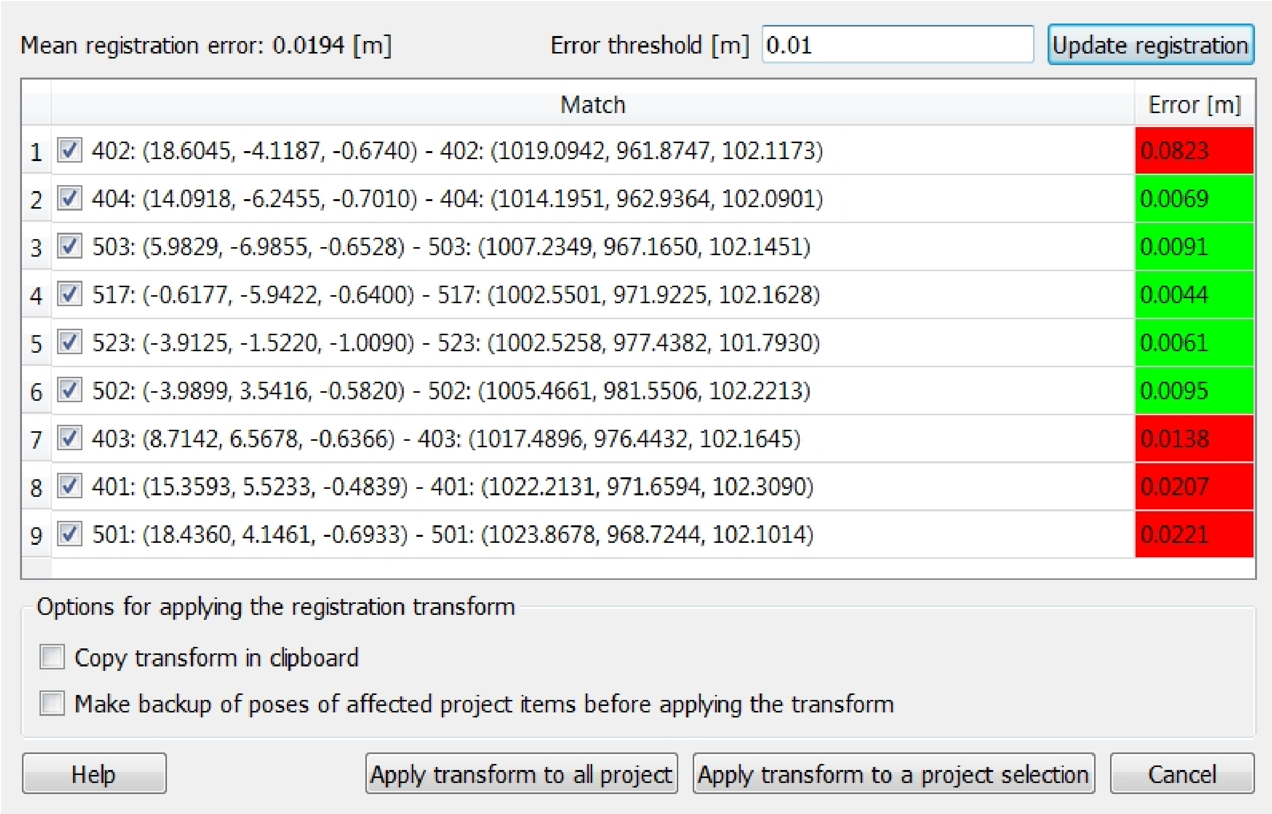


This dialog allows you to register a list of points, taken from a text file or from the point list window, against another list of points that you consider as reference. In particular, these last points can be *geo-referenced.*
The left half of the dialog regards the *moving* points. You can load them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

This tool registers a list of points against another list of points. For a general overview of JRC 3D Reconstructor®'s registration tools, see Registration techniques.

Registration report dialog

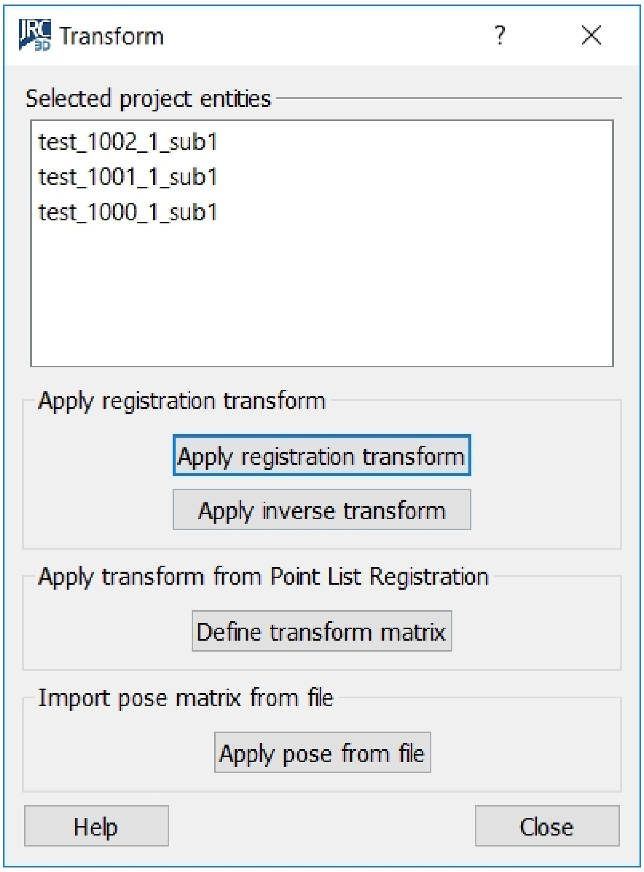
Registration report dialog



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.*
- 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.

Pose Transform



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 A*pply registration transform* and D*efine 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 JRC 3D Reconstructor®'s project entities,
- *Define transform matrix* is suited for transforms coming from other tools, or transforms calculated between entities not belonging to JRC 3D 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 JRC 3D 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 *Apply 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 registraton 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 JRC 3D 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, JRC 3D 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.

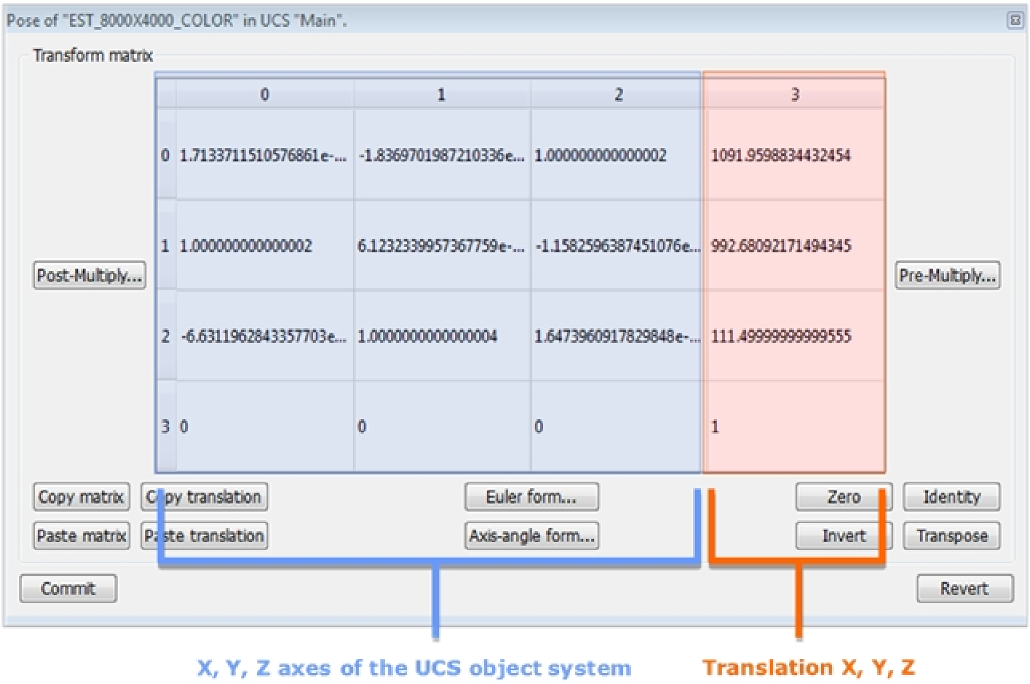
See also Pose Dialog.

Pose Dialog

In JRC 3D Reconstructor®, the 3D position of each object (except annotations) is described by the POSE matrix (*transformation matrix*) that you can open pressing the POSE button in the Property Browser Window or by using the contextual menu of items (*Pose&Registration->Pose*).

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.



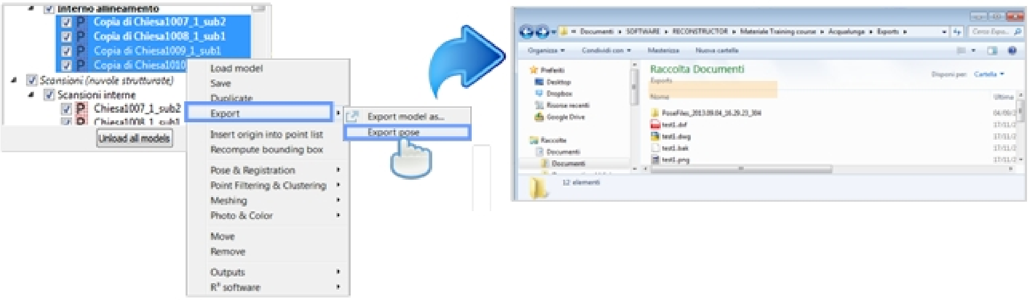
A *transformation matrix* is a 4x4 matrix that specifies how an object *O* is located and oriented with respect to a global coordinate system. The first, second and third column of the transformation matrix contain respectively the versors of the *X*, *Y*, and *Z* axes of the system anchored with the 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

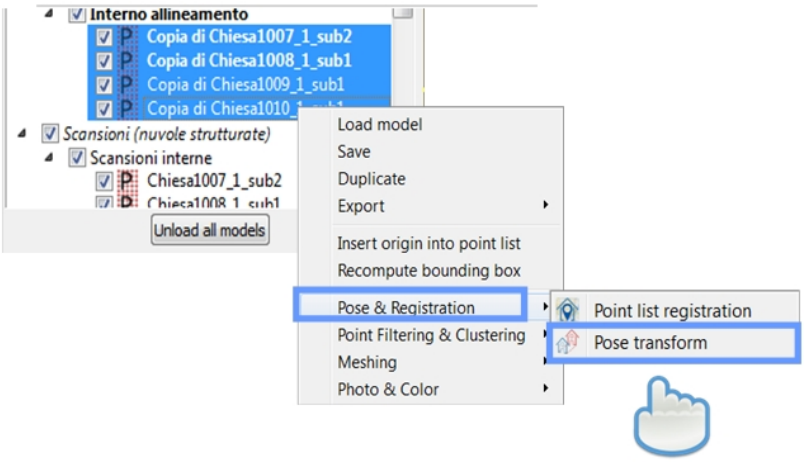
You can restore an object position in case of registration and geo-referencing error once the Pose file is saved.

To save the Pose you have to export it:

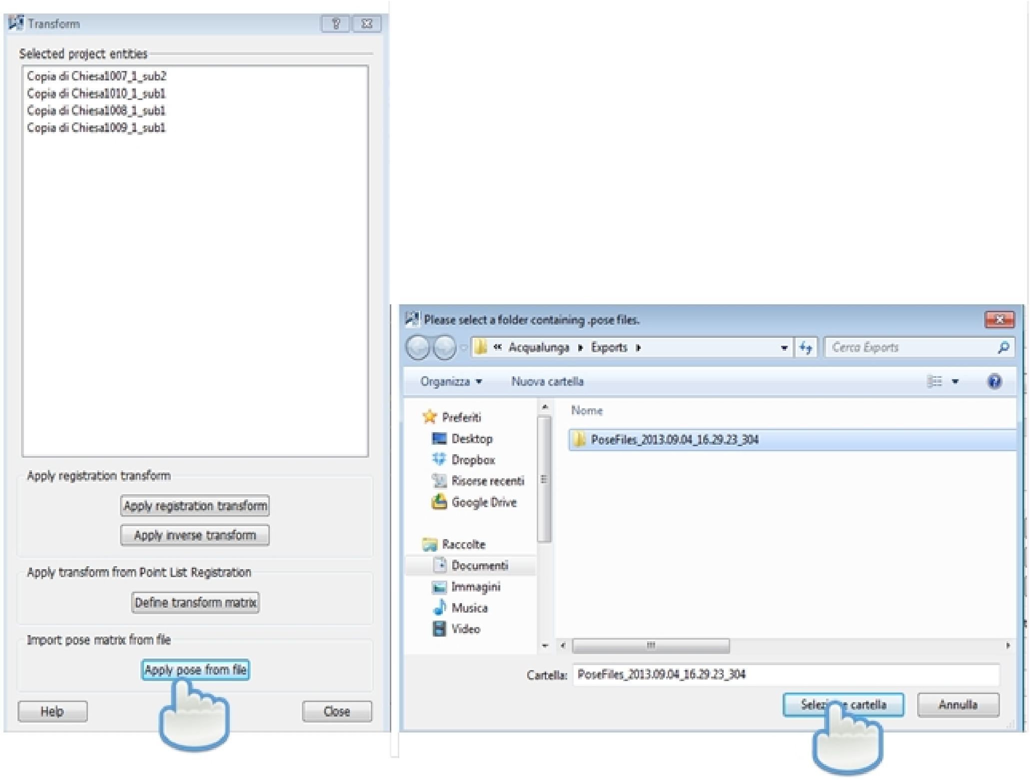


The software automatically creates a folder *PoseFiles_2013.09.04_16.29.23_304* that contains the Pose exported. This folder is saved in the Exports folder and the name contains the date (day and time) of the creation.

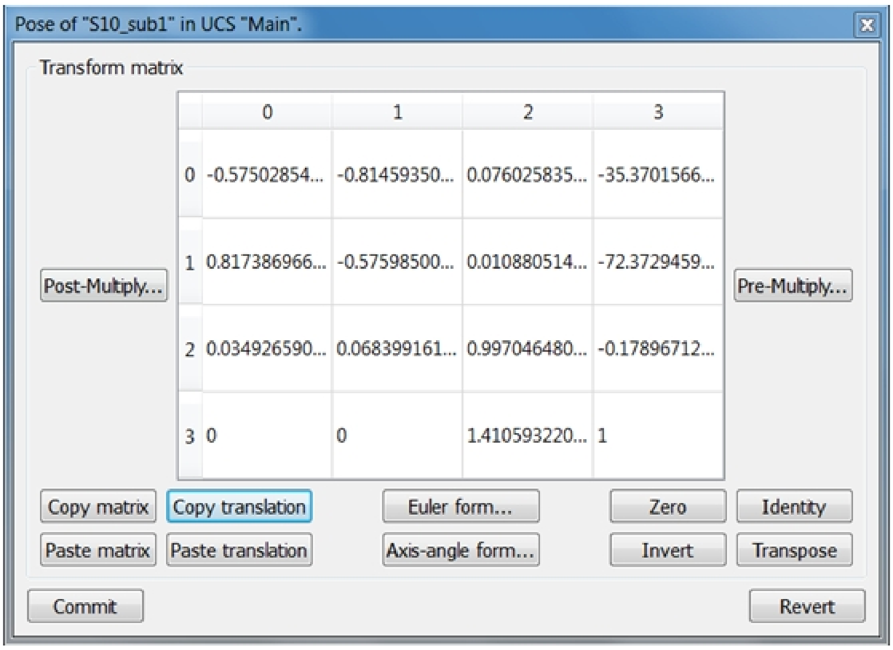
Apply the Pose file by selecting the objects desired → right mouse button → Pose&Registration → Pose transform.



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



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

1	0	0	0
0	0	-1	0
0	1	0	0
0	0	0	1

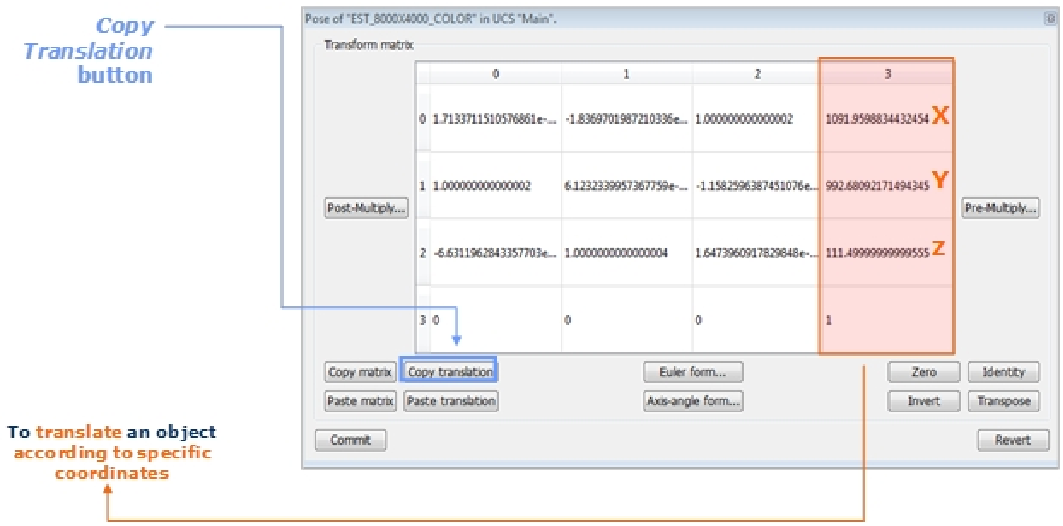
Rotation around Y axis by 90 degrees

0	0	1	0
0	1	0	0
-1	0	0	0
0	0	0	1

Rotation around Z axis by 90 degrees

0	-1	0	0
1	0	0	0
0	0	1	0
0	0	0	1

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.

Selection & Fitting Tools

In this section you can learn how to select portions of point clouds the point clouds and how to fit it with some features.

- Select
- Point selection with polyline
- Sample to new cloud
- Fitting

Select

In this section you can learn how to select portions of one or more point clouds.

You can find the command *Select* in *Tools-> Selection and fitting tools* or in the upper right side of the Top tool bar.

You can switch between navigation and selection mode by pressing the spacebar.

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

Rectangle
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.

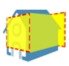
Polygon
To make a polygonal selection of the point clouds in the current view.
click LMB for each point of the polygon. Close the polygon by double clicking the LMB.

Lasso
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*

During points selection you can choose how select points:

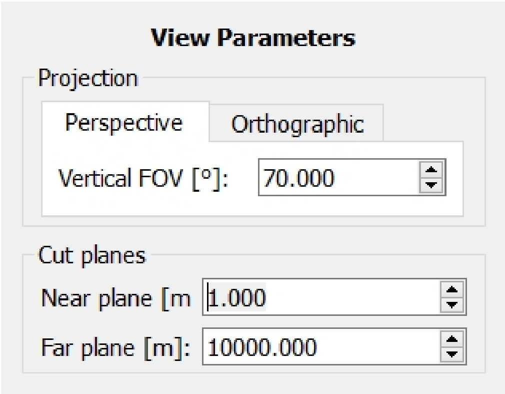


Deep Selection
To sample the selected region. All points that fall into the current video selection, even if they are hidden by more prominent objects, are selected.

View Selection
To sample the selected region from the depth buffer. This means that only the visible points will be selected. The accuracy of the points depends on the current frustum depth range. The

number of points is exactly the one of the current screen resolution.

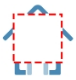
Suggestion: get close to the object to select and keep the maximum order of magnitude between near and far plane in view cut planes under 4.



[See View Parameters Dialog]

Reset samples
To discard all the current samples.

There's two ways to delete points just selected:

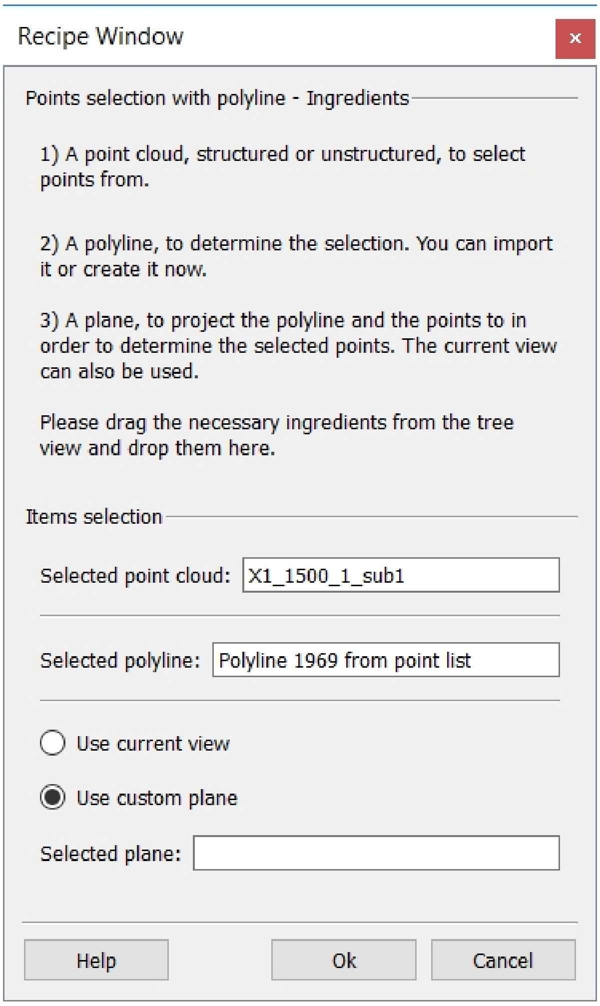


Delete inside
The points that fall inside the selected region are deleted from the view. They can be undeleted at a later moment.

Delete outside
The points that fall outside the selected region are deleted, and can be undeleted at a later moment.

See also Restore Deleted Points to learn how to undelete points.

Point selection with polyline



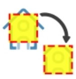
This is a tool for selecting points using a 3D shape, in particular a polyline.

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.

Samples to new cloud

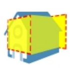


You can find the command *Samples to a new cloud* in *Tools-> Selection and fitting tools*.

The management of a great project, formed by a large number of scans, can be very difficult.
When it's possible to work on a small region it's more advisable to extract a point cloud's portion to lighten the processes.

Starting from a selection of points (see Select), the command *Samples to a new cloud* creates a new point cloud called *Sample* and stores it as a new item in the project.

Which type of point cloud?



If you selected the points with *Deep Selection* command a new Unstructured point cloud is created.

If you selected the points with *View Selection* command a new Grid (structured) point cloud is created.

Selection and fitting

After selecting point samples with Selection Tools a surfaces fitting is possible.

Fit plane
Fit a plane to the current samples and add it to the project

Fit cylinder
Fit a cylinder to the current samples and add it to the project

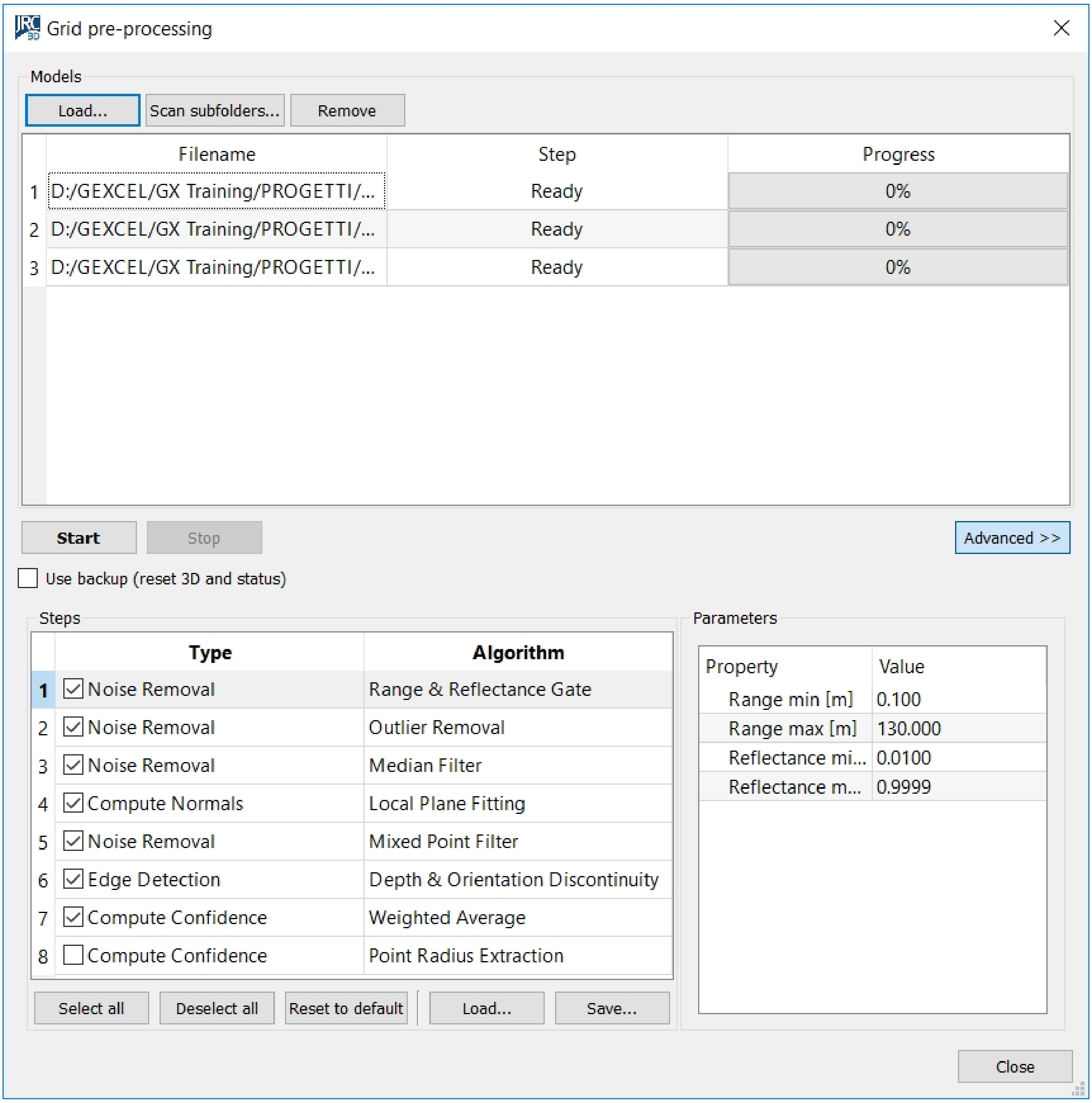
Fit sphere
Fit a sphere to the current samples and add it to the project

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 cloud, some of them work only on grid point clouds.

- Pre-process Grids
- Restore raw data
- Restore deleted points
- Edit 2D (Grid point clouds only)
- Level 3D density of clouds
- Make single cloud
- Fill holes (Grid point clouds only)
- Hide black points
- Remove duplicate points
- Resample
- Simplify points (Grid point clouds only)
- Extract edges (Grid point clouds only)

Pre-processing Grids



During preprocessing JRC 3D 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 contextual 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*.

JRC 3D 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.

To modify the pre-processing parameters press **Advanced**, then select one or more rows in the list of models with Ctrl or Shift, then select a *step* of processing. The parameters are updated for all the selected models. 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.

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.

Median Filter

Mask border [pix]: the kernel of the filter is a square mask (of side 2·border+1) centered at the current pixel.

Mixed Point Filter

Useful as a relative depth discontinuity filter. Mask border [pix]: the kernel of the filter is a square mask (of side 2·border+1) centered at the current pixel.
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.

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.

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.

Restore Raw Data

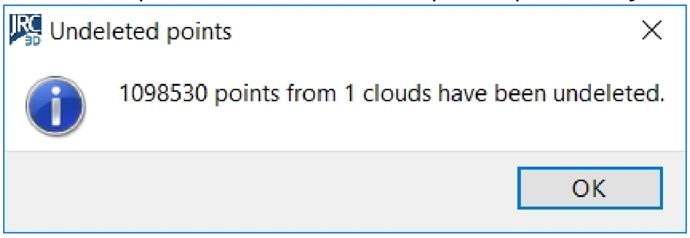
Input data: at least one unstructured point cloud must be selected.
This functions takes as input a set of point clouds and undoes any operation of preprocessing, deletion and editing that may have been performed on the clouds. Its main usefulness is to guarantee that you never lose data: if you are 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.

Therefore, when applying this function:

- All deleted points are visible again
- Points “confidence” layer is deleted
- Points' normals are deleted
- 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

Restore deleted points

Sometimes happens that any point are deleted for mistake.
In JRC 3D Reconstructor® there's a method to recover deleted points, even in different times, valid for both Grid and Unstructured point clouds.
The command *Tools* → *Points Filtering&Clustering* → ***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 command *Predefined Views* → *2D view* (also in the contextual menu of the grid *Filtering&Clustering* → *2D view*) 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.

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. If the AutoCAD link is enabled, the 3D coordinates are sent to AutoCAD
- **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 polylin

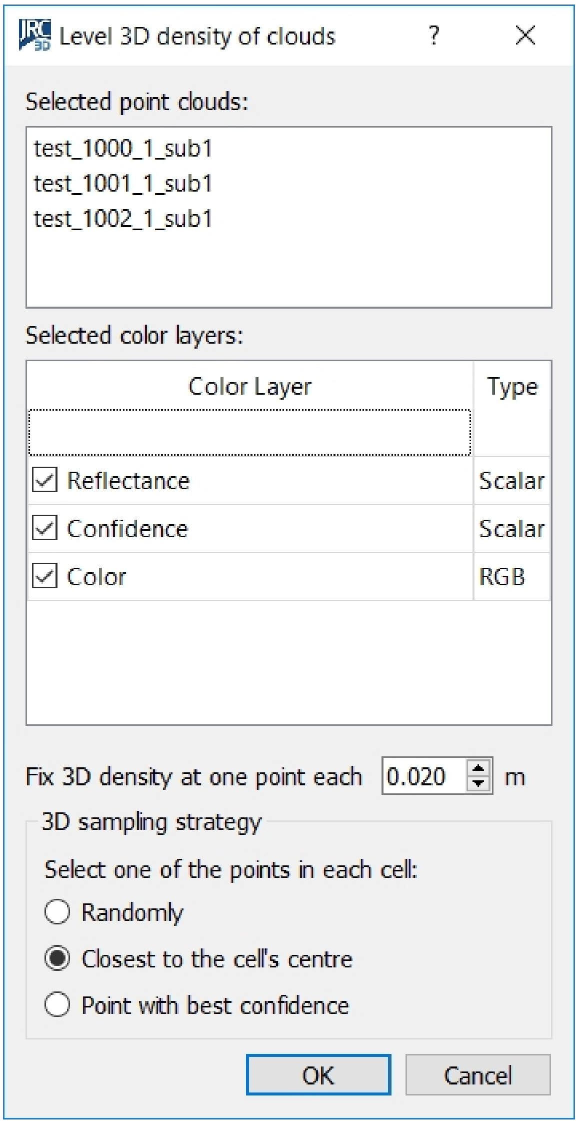
Edit operation

- **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 untouche

Show

- **Color type**: display the grid with the selected color type. Press the right arrow to show a menu of available commands for the current color type
- **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.
- **Global coordinates**: display the 3D global coordinates of the point (i.e. using the pose of the model). Otherwise, the local model coordinates are used.
- **Save image**: save the current image to file.

Level 3D density dialog



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.

To activate this dialog, you can select a set of point clouds from the project window and select *Point Filtering&Clustering→Level 3D density of clouds* from the contextual menu or from the tools menu.

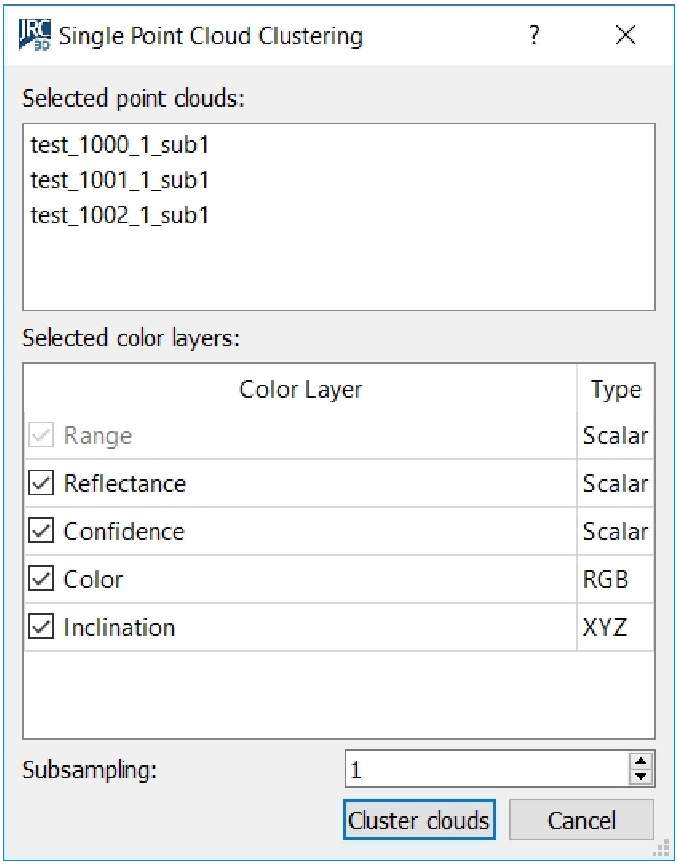
The selected clouds appear in the top list of the dialog. Below, the common color layers across the clouds are found. The cloud that results from the clustering will contain only the color layers that are present in all input clouds and that are chosen. Point normals will be transformed so that they 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.
This procedure will efficiently exploit all CPU cores of your PC.

Make single cloud

(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.

To activate this dialog, you can select a set of point clouds from the project window and select *Point Filtering&Clustering->Make single cloud* from the contextual menu or from the tools menu.

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.

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 neighbourhood in the cloud's structure. After activating this tool, Reconstructor asks you the mask border M: the search neighbourhood 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 unstructured point cloud must be selected.

This functions takes as input a set of point clouds, for each of them takes the current color and invalidates all the points in the cloud that are colored in full black (RGB components [0 0 0]).

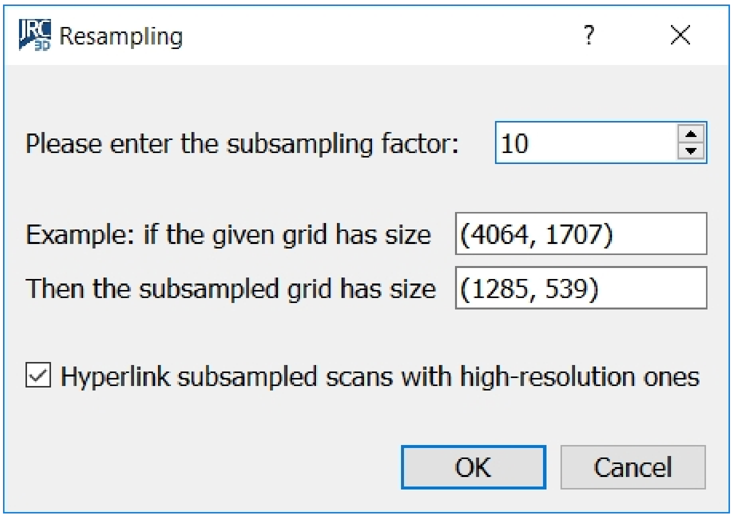
Remove duplicate points

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 3D 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 JRC 3D 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.

Resampling dialog



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.

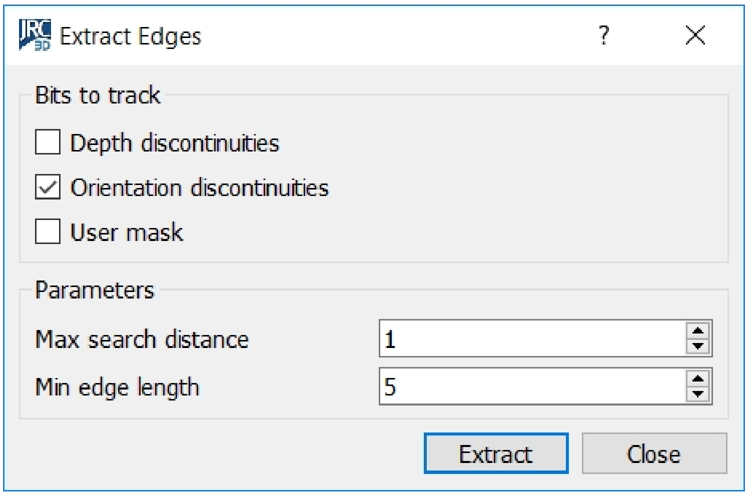
Simplify points

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 dialog



This tool allows you to extract the edges of a grid point cloud, in form of polylines. You can activate this function from the top menu by clicking *Tools*->Point *Filtering & Clustering->Extract edges*, or by the contextual menu of any grid point cloud.

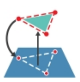
You can select which discontinuities in the 3D point cloud you want to mark as edges. These can be the *Depth discontinuities* (points that are neighboring in the grid structure but far away in 3D position), the *Orientation discontinuities* (points that are neighboring in the grid structure, close to each other in 3D position, but having normals very different), or both. You can also mark as edges all points previously marked with the *User mask*. This is a mask reserved for advanced operations, for example the *mask filter*.

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.

Meshing

In this section you can first learn how to create triangle meshes.

JRC 3D 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	✓ (one)	✓ (one – single or clustered	Useful for façades (using orthocamera) and tunneling (using cylindrical camera)

	a vertex)		point cloud)	
	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

(*) Follow links for procedure details.

These commands are achievable from *Tools→Meshing*, from *Mesh tools* in the top toolbar or from the context menu of point clouds.

See the section Mesh Editing to learn all the editing meshes techniques.

Multiresolution Meshing

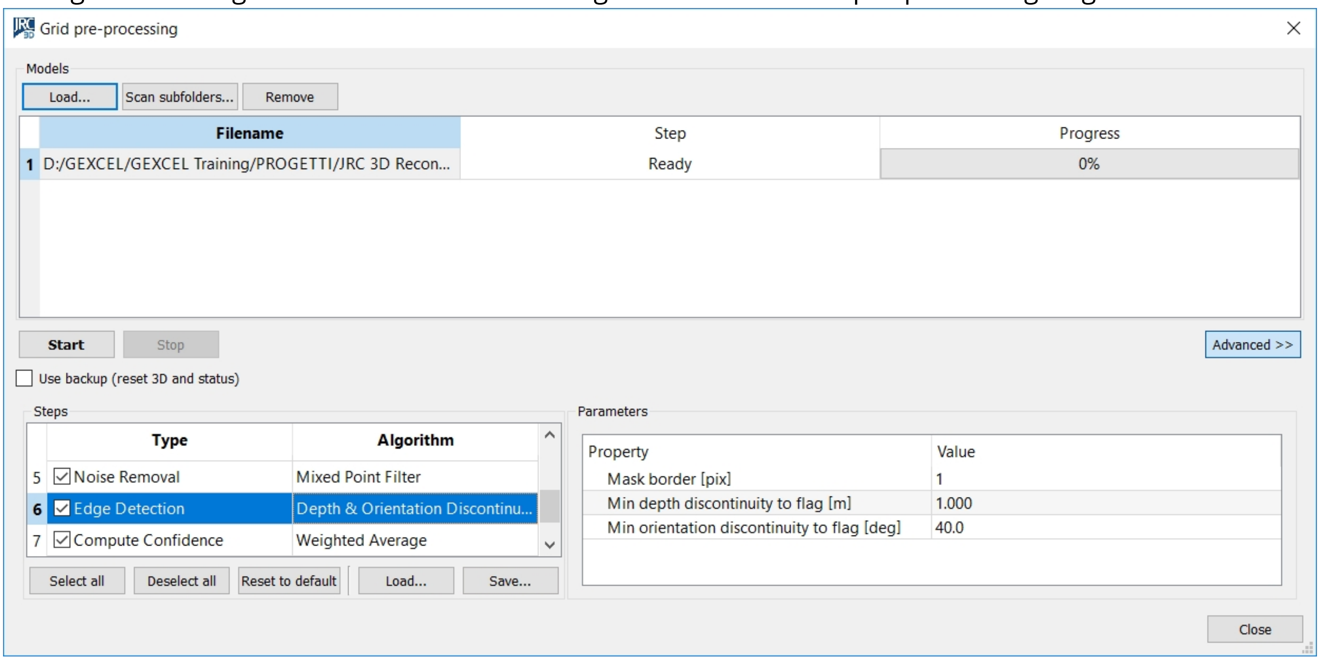
Multiresolution Meshing is possible only with Grid point clouds. If you want to generate a mesh from an Unstructured point cloud, first transform it in a Grid point cloud by using:

1. the Virtual scan tool
2. the *Tools→Selection and fitting tools→ Samples to a new cloud* after selecting the unstructured points with Selection tools and the *View Selection* command

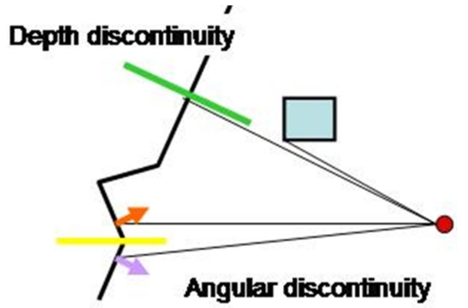
NOTE

Before meshing you need to Preprocess the Grid.

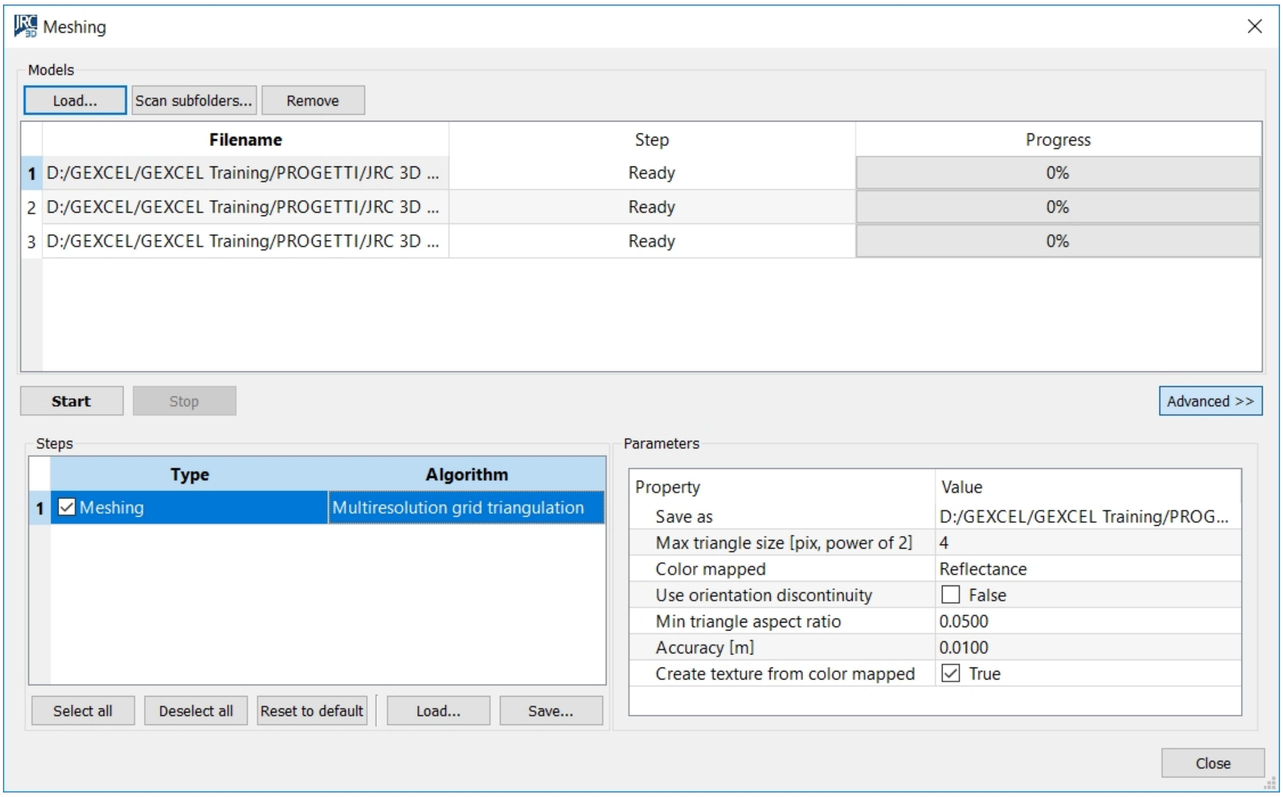
During the meshing the software will use the Edges calculated at the pre-processing stage:



- *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.

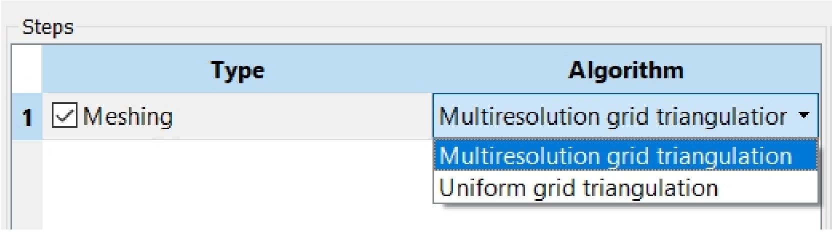
- Press **Start** to run the processing. This plug-in creates a separate thread for each grid so if multiple CPUs are available the processing is sped up proportionally.

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.

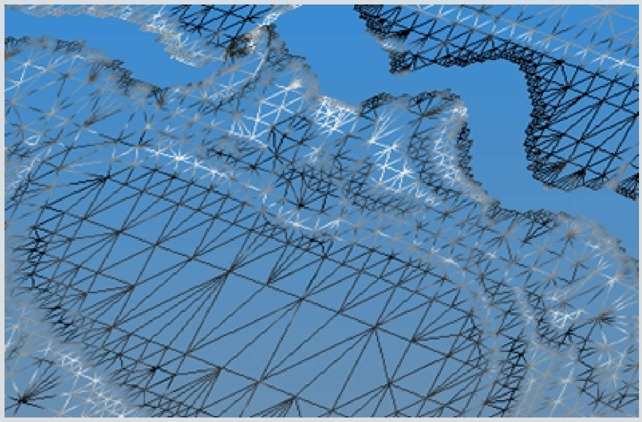
[Note: Aspect ratio of a triangle is the ratio of the inradius to its circumradius). The aspect ratio of a triangle lies between 0 and 0.5. For triangles with an angle near to zero, the aspect ratio is 0; for equilateral triangles, the aspect ratio is 0.5.]

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.

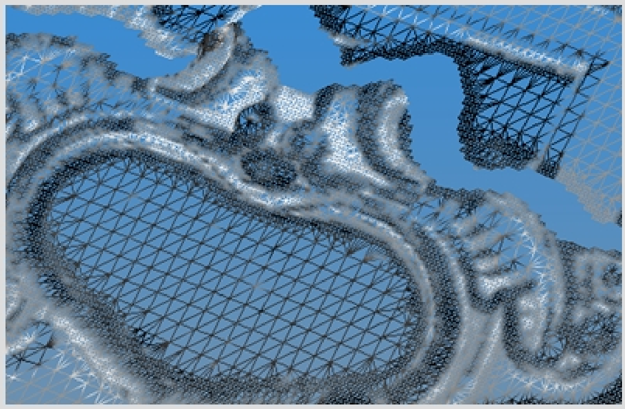
In the images below an example of the use of two different set of parameters is showed.

Save as	C:/Users/Rossi/Documents/SO...
Max triangle size [pix, powe...	4
Color mapped	Color RGB
Use orientation discontinuity	☑ True
Min triangle aspect ratio	0.0500
Accuracy [m]	0.0100
Create texture from color m...	☑ True



Accuracy ↓
Max triangle size ↑
Use orientation discontinuity: TRUE

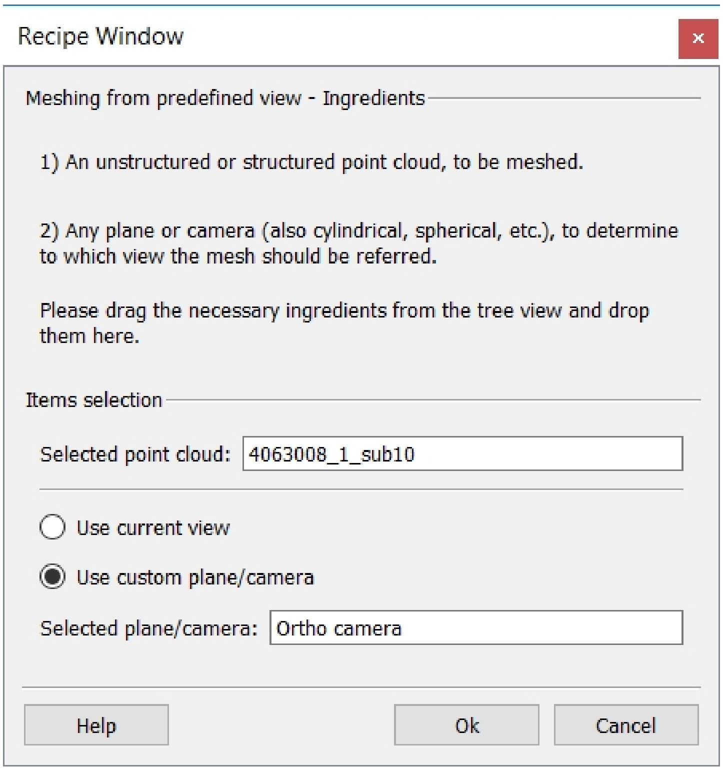
Save as	C:/Users/Rossi/Documents/SO...
Max triangle size [pix, powe...	2
Color mapped	Color RGB
Use orientation discontinuity	☑ True
Min triangle aspect ratio	0.0500
Accuracy [m]	0.0080
Create texture from color m...	☑ True



Accuracy ↑
Max triangle size ↓
Use orientation discontinuity: TRUE

See also other Meshing techniques.

Meshing from predefined view

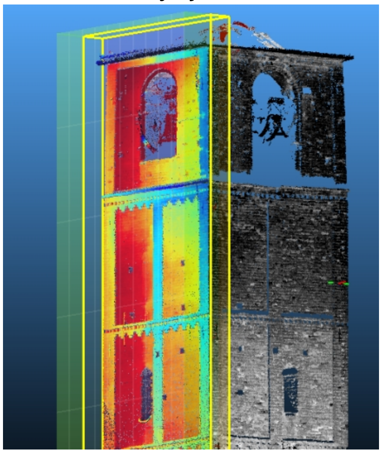


This is a meshing technique that provides watertight meshes. It assumes that the desired mesh is such that each point of it has one and only one projection according to a *predefined view*. In other words, it assumes that a perspective, orthographic, cylindrical or spherical projection exists such that the desired mesh never hides parts of itself if seen from that projection.

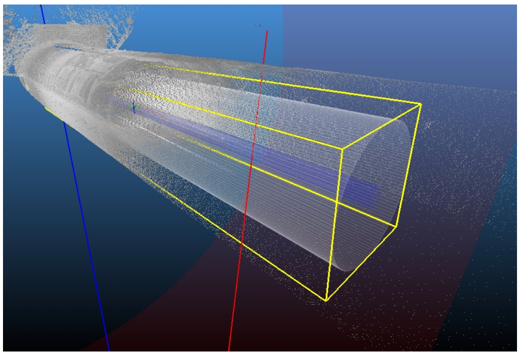
In order to create the mesh you need:

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. JRC 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.

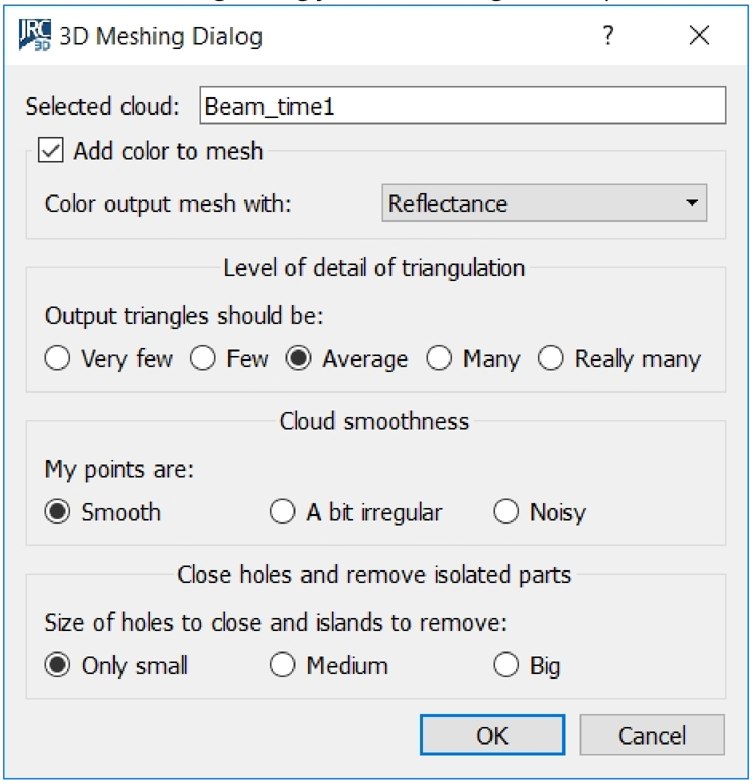
3D Meshing (Poisson reconstruction)

After selecting point samples with Selection Tools a surfaces fitting is possible.

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 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.

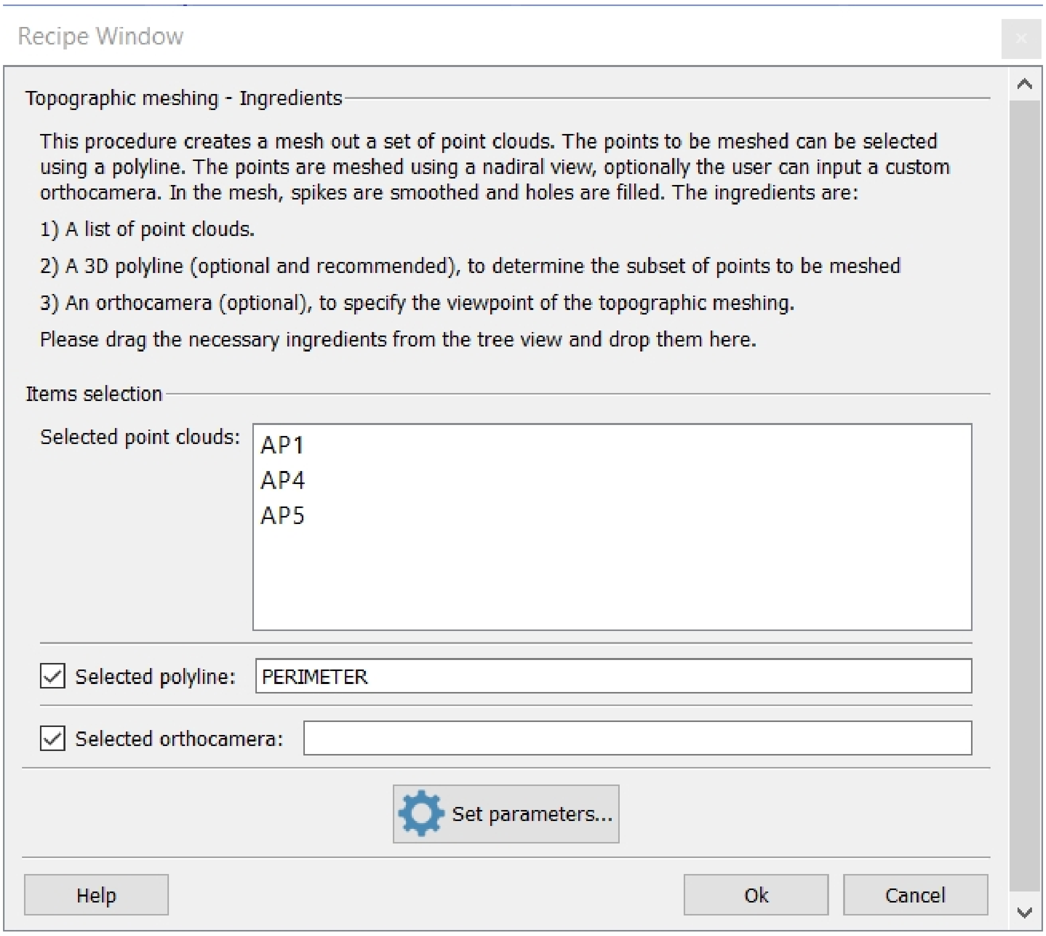
In the 3D Meshing Dialog you can manage some parameters to obtain a best fitted mesh.



Adding color
In the first parameter box of the dialog, you can select whether the output mesh should be colored or not, and which of the input cloud's color layers should be used.
Specifying how many details
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 triangles you want to use for your output mesh. The more the triangles, the finer the details that will be reconstructed, and also the heavier the mesh file and rendering time.
Taking into account cloud smoothness
In the third parameter box, you can take into account the cloud smoothness. If your cloud represents clean construction data, then select *Smooth*. If, however, your cloud contains noisy territorial data, then select *Noisy*.
Closing holes and removing isolated components
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. The size is computed as ratio between the area of the hole/island and the area of the whole mesh.

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:

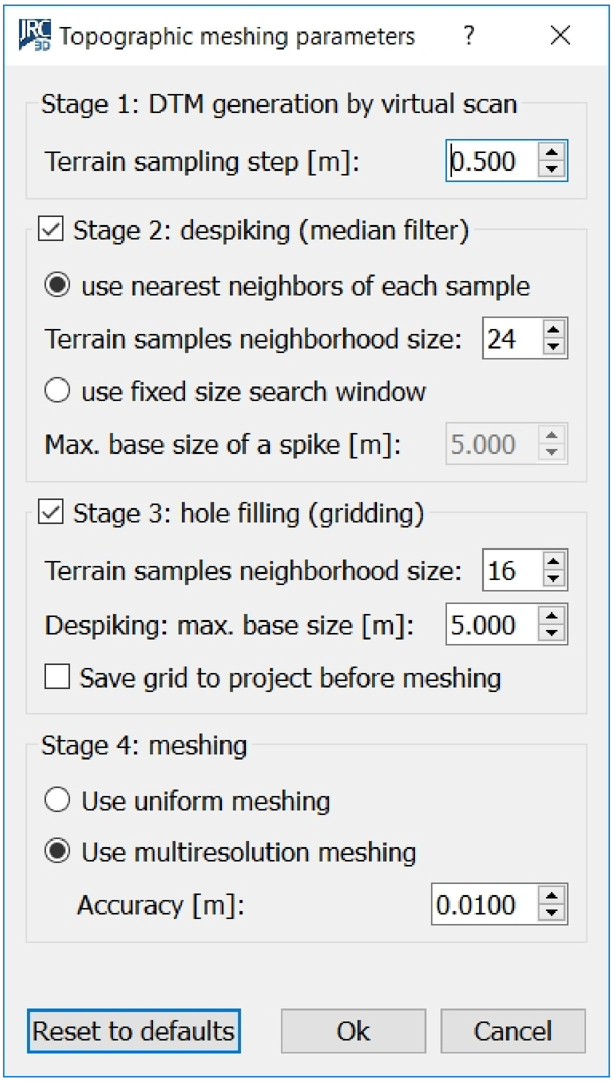
- Virtual scan is performed from the (nadiral) orthocamera, to uniformly sample the point clouds.
- Then, spikes in the resulting samples are smoothed using a median filter based either on a neighborhood of given size, or on a window of fixed size.
- Successively, a polynomial interpolation (ordinary 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.

See also other Meshing techniques and Topographic meshing parameters.

Topographic meshing parameters



In this dialog you can set the parameters of the topographic meshing procedure.

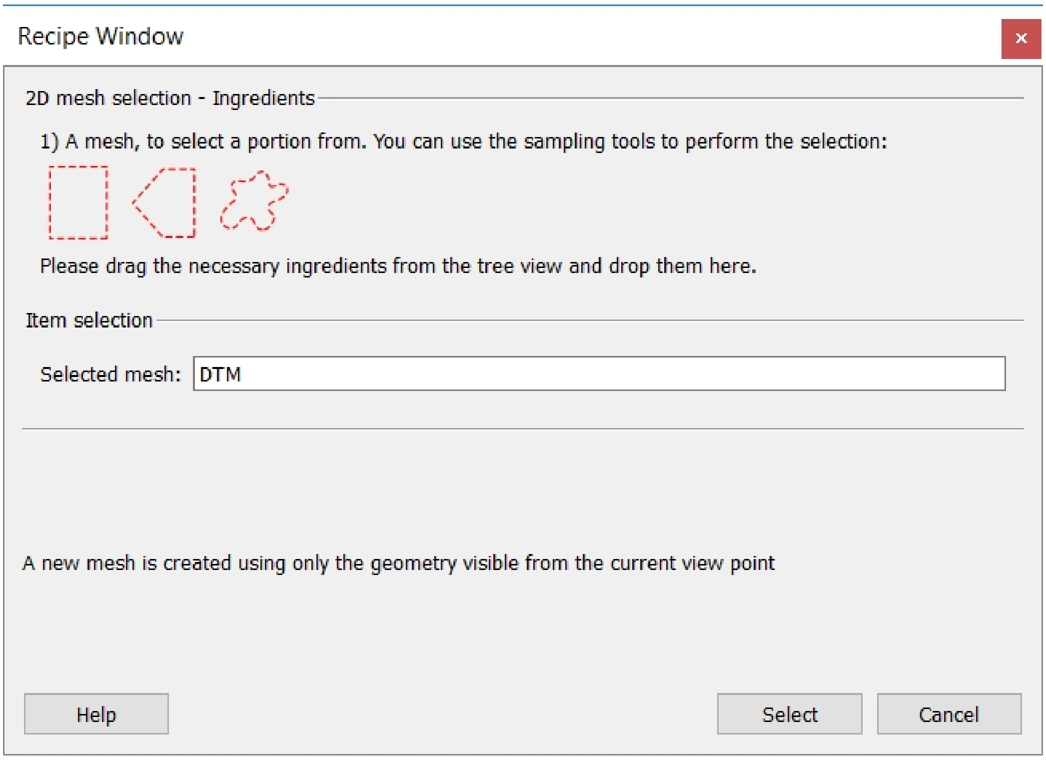
This dialog shows from top to bottom the four stages of the topographic meshing procedure. You can customize the parameters of each stage, and even decide to skip stages 2 and 3 by unchecking the respective boxes.

- **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.

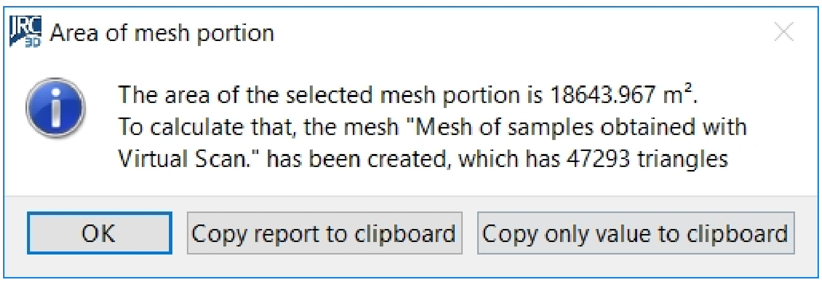
Use the *Reset to defaults* button on bottom left if you are not sure of what is happening. However, topographic meshing is so fast that it is worth to try many combinations of parameters and compare the results.

Mesh selection from current view point



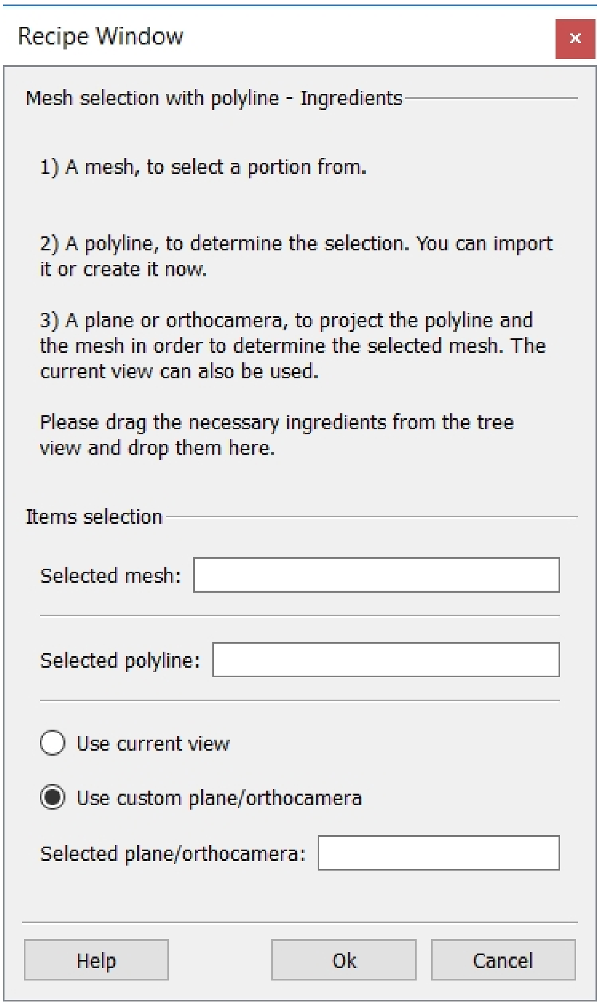
This function allows to cut a portion of a mesh using 2D video selection tools on the current view. It is accessible through *Tools-> Meshing->Mesh selection from current view point*. When you activate this function, the window above appears in the recipy window, and JRC 3D Reconstructor®'s 3D window 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.

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 JRC 3D 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, JRC 3D 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.

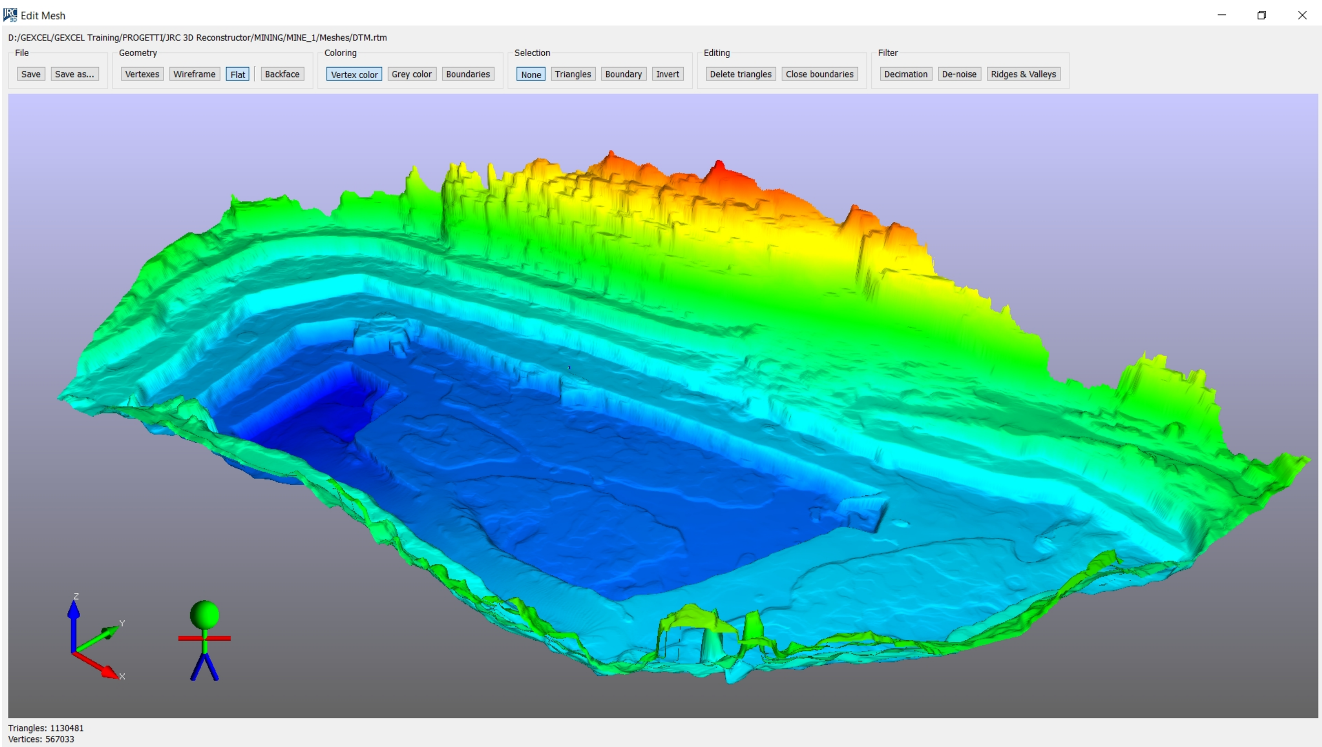
Mesh Editing

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.

JRC 3D 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, ridges and valleys 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
Make single mesh...	This dialog allows to lump together in a single mesh an arbitrary set of triangle meshes
Convert 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
Compute 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 *Tools->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.

Top toolbar

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 JRC 3D 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

The first two buttons here 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. When you are drawing the selection rectangle, keeping pressed *Shift* will enable you to add triangles to the current selection by drawing another selection rectangle. 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.

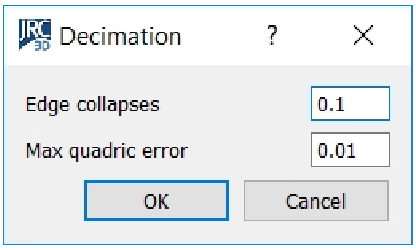
Editing

Once you have selected some triangles with the button *Triangles* explained in the former paragraph, you can delete the selected triangles with *Delete triangles*. The button *Close boundaries* allows you to close a mesh hole whose boundaries have been selected. This function works best with small and with close-to-planar holes.

Filter

This button group contains three functions:

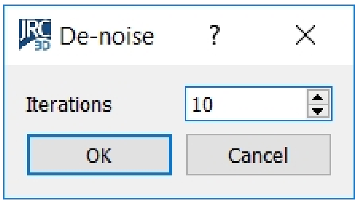
- *Decimation*



This 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.

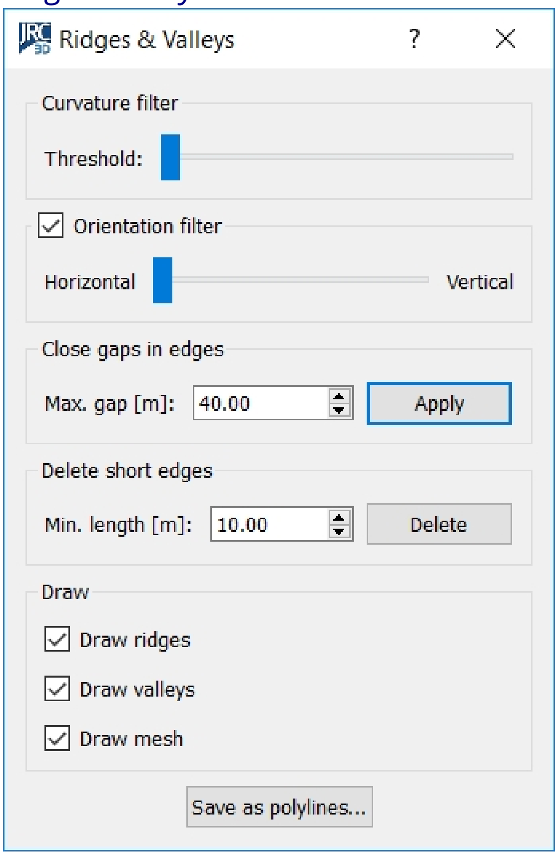
For each edge shared by two triangles, JRC 3D Reconstructor® assesses whether collapsing it or not. Collapsing an edge means assigning the edge's two vertices to coincide in the edge's middle point, and adjusting all neighboring triangles consequently. If the quadratic error between the surface before 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%.

- *De-noise*



This 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.

- *Ridges&Valleys*



In some workspaces, this can be called *Crests&Toes*. This is a semi-automatic technique to detect Ridges (prominent mesh edges) and Valleys (reentrant mesh edges). When you open this dialog, the ridges and valleys are calculated and drawn in the mesh editor. Ridges are drawn as blue polylines, valleys are in red. Ridges and valleys can be then saved as polylines.

The idea is that tool shows you an initial extraction of ridges and valleys, 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 ridges and valleys as polylines in the project. The parameters of ridges&valleys 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 ridges&valleys, you can choose to turn on or off the visualization of the ridges, the valleys 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 treshold on the *curvature* of the ridges and valleys. Ridges and valleys 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 ridges and valleys 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 ridges and valleys 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.

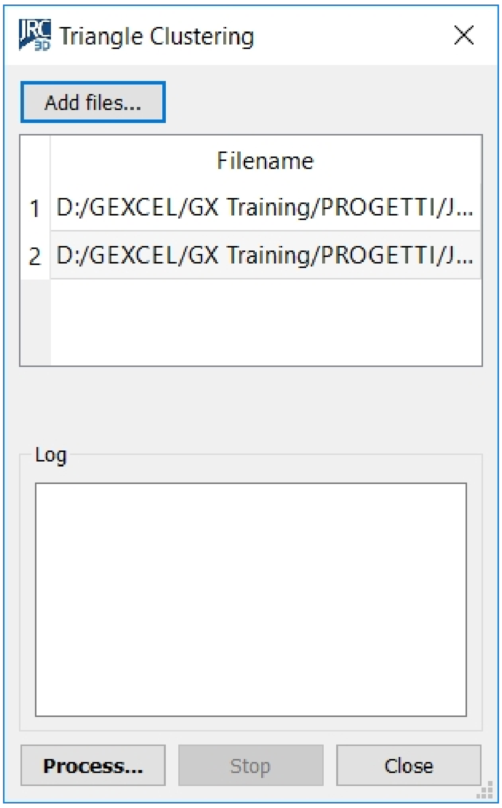
Delete short edges

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 ridges and valleys 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 ridges and valleys 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 clustering tools and triangle meshes.

Photo & Color

JRC 3D 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

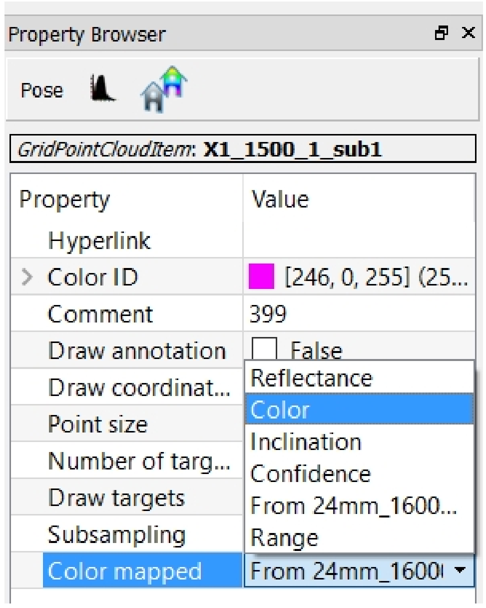
 ***Histogram***	To optimize the point clouds color's contrast by histogram stretching
 ***Colors mapping***	To create an artificial colorization for a given *color layer* of a point cloud. Color information can be scalar (e.g. reflectance, range, confidence) or vectorial (e.g. inclination). It's also 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 point 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. *Suggestion*: use it to classify points for a further vegetation removal.

- Some color information is due to enbedded cameras or external GEOtiff:

	Create Projector	To create a Perspective, Orthographic, Spherical , Cylindrical projector or from a calibration (by importing a JRC 3D Reconstructor® camera calibration file) using a valid image.
	***Import georeferenced tif***	This function imports a geo-referenced TIFF as an orthographic camera.

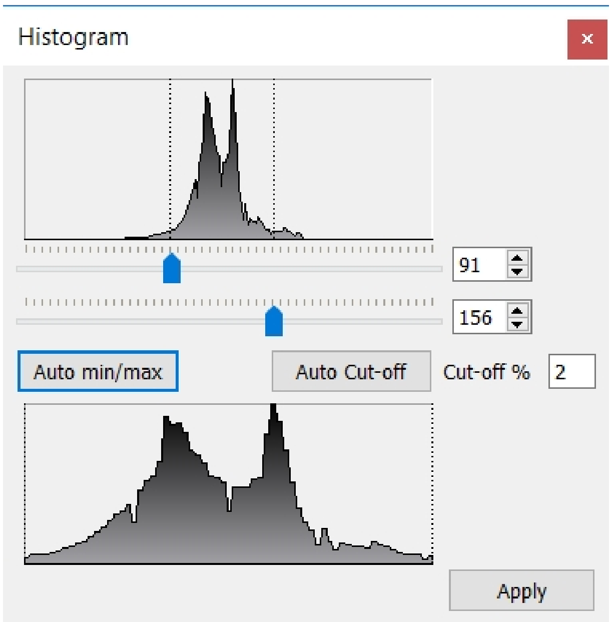
The coloring tools are accessible from the *Tools->Photo&Color* menu or from the point clouds' contextual menu *Photo&Color*. See also *Photo and Color* in the Top Toolbar.

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.



See also our video tutorials on point clouds coloring.

Histogram dialog



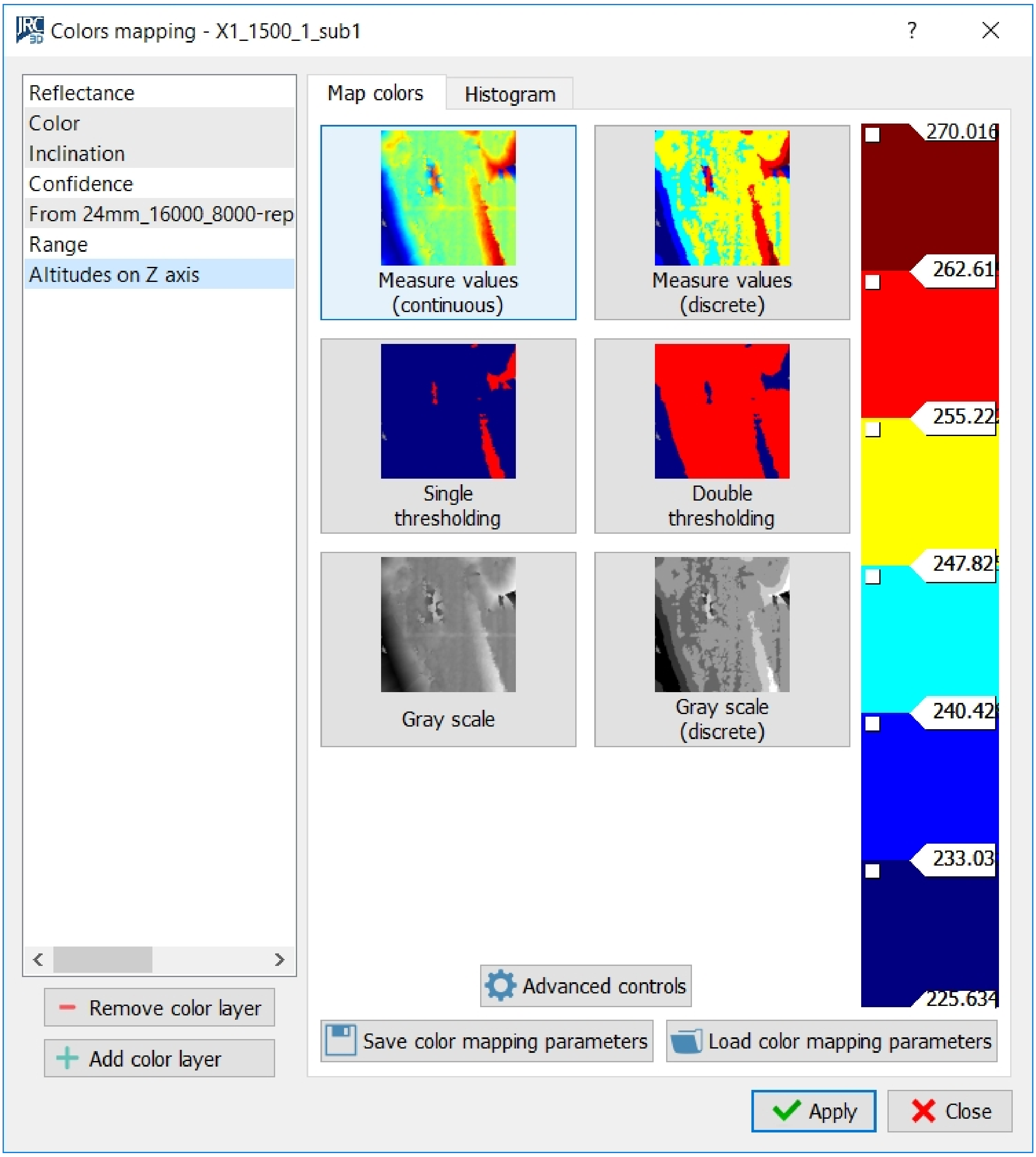
Given a point cloud and one of its scalar color layers (e.g. Reflectance), the histogram tool allows to optimize the color's contrast by histogram stretching. The histogram shown in the dialog represents how many points belong to a given color value.

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.

Colors mapping

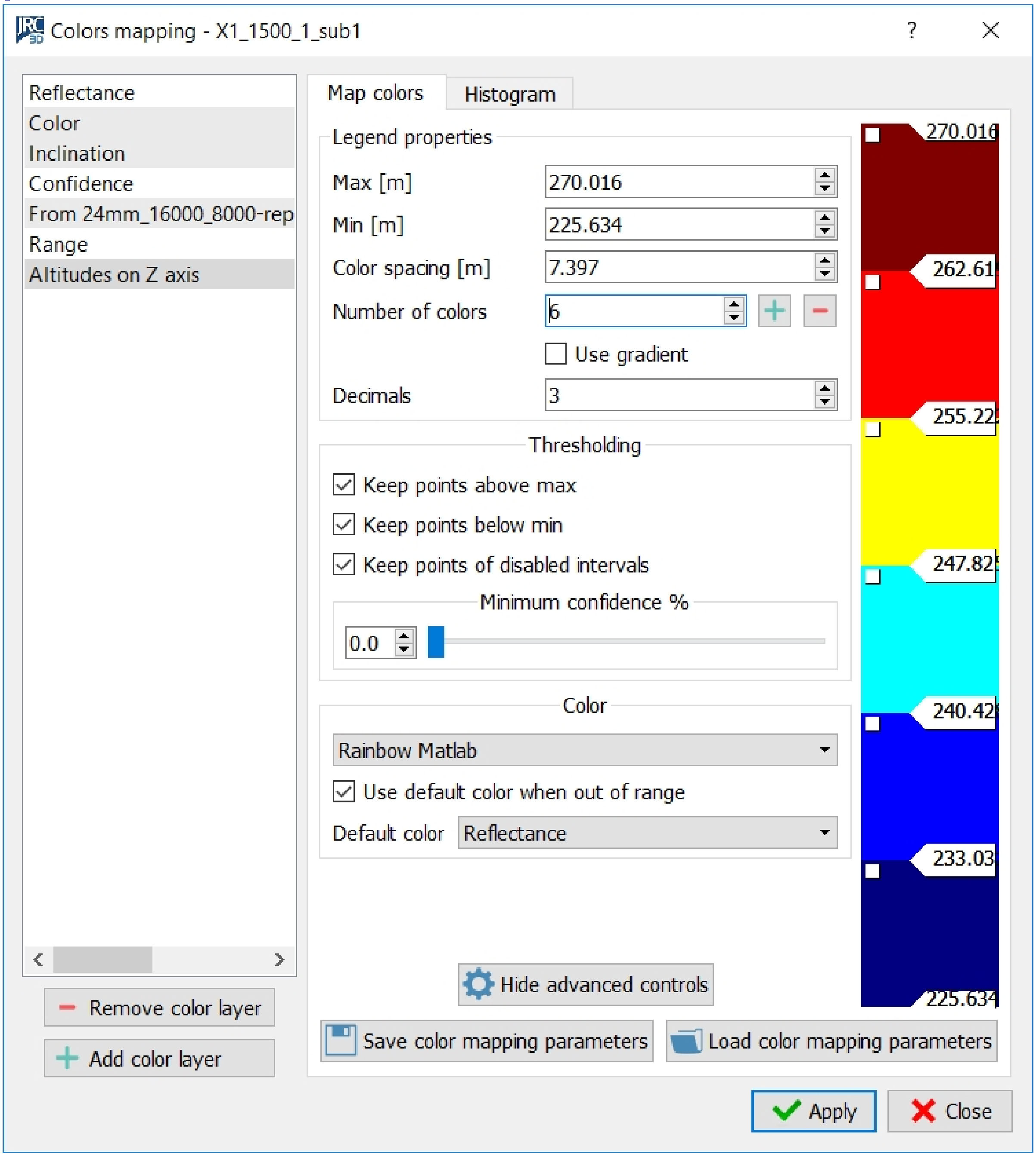
The purpose of this command is to manage and to create an artificial colorization for a given *color layer* of a point cloud.
The command is achievable from the menu *Photo&Color* both on the Tools Menu bar and in the Top toolbar.



The user can manage the rest of JRC 3D 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, error bound, confidence) or vectorial (e.g. inclination). User can add color layers to a cloud via functions like *Inspection* or *Color with altitudes by clicking Add color layer tab or* from the menu .
Only one color layer at a time is rendered in the 3D scene. User can change a cloud's current color from the layers list on the left of the colors mapping dialog or by clicking on property browser. The change take place after pressing the *Apply* button.

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.

Map colors tab



Layers list

On the left there is a layers list that show which layers are associated to the current point cloud.

Predefined colors buttons

On the Center there is a set of buttons to apply a pre-defined colorizzation 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

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 *Predefined buttons* or by clicking on *Advanced button*. User can also disable colors by clicking on the color check box.

Advanced controls

The user can master colors and design or produce to meet his or someone's individual requirements. Advanced commands are divided in three groups.

Legend properties:
- *Max* to set the max value and show it in the legend
- *Min* to set the min value and show it in the legend
- *Colors spacing* that give information about width of the color
- *Number of colors* that is useful to set the amount of colors to use
- *Use gradient* to switch between a discrete and a continuous colorization
- *Decimals* to set how many decimal digits use

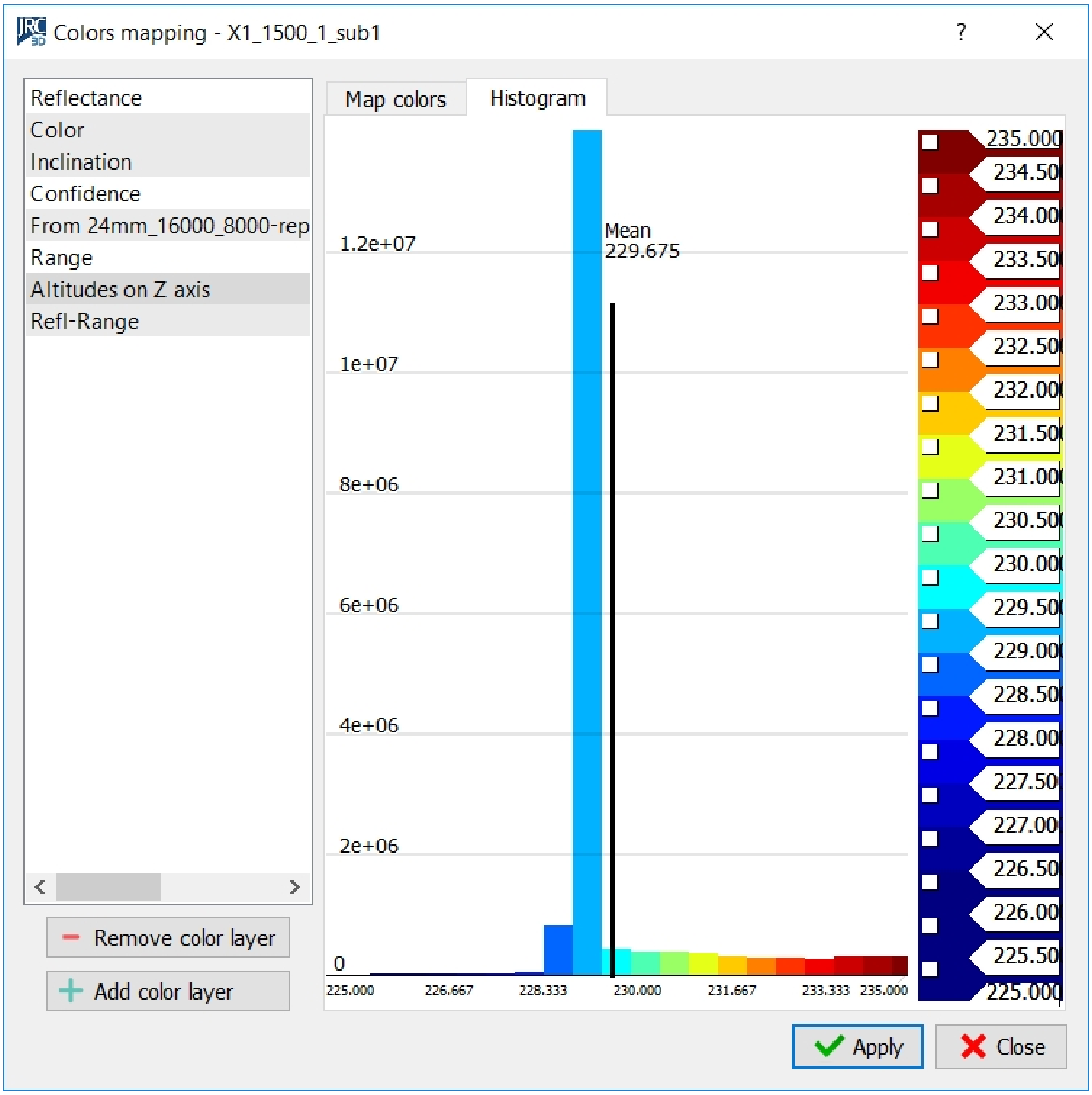
Thresholding group:
- *Keep points above max* if selected keep points above *"Max"* value else points are hidden.
- *Keep point below max* if selected keep points below *"Min"* value else points are hidden.
- *Keep point of disabled intervals* if selected keep points in the deselected color else points are hidden.
- *Minimum confidence* to set manage confidence value.

Color group:
- *Combo box* to chose between six color styles:
1. Single thresholding
2. Double thresholding
3. Extended
4. Gray scale
5. Rainbow
6. Rainbow Matlab
7. HSV scale
8. Customize

- *Use default color when out of range*

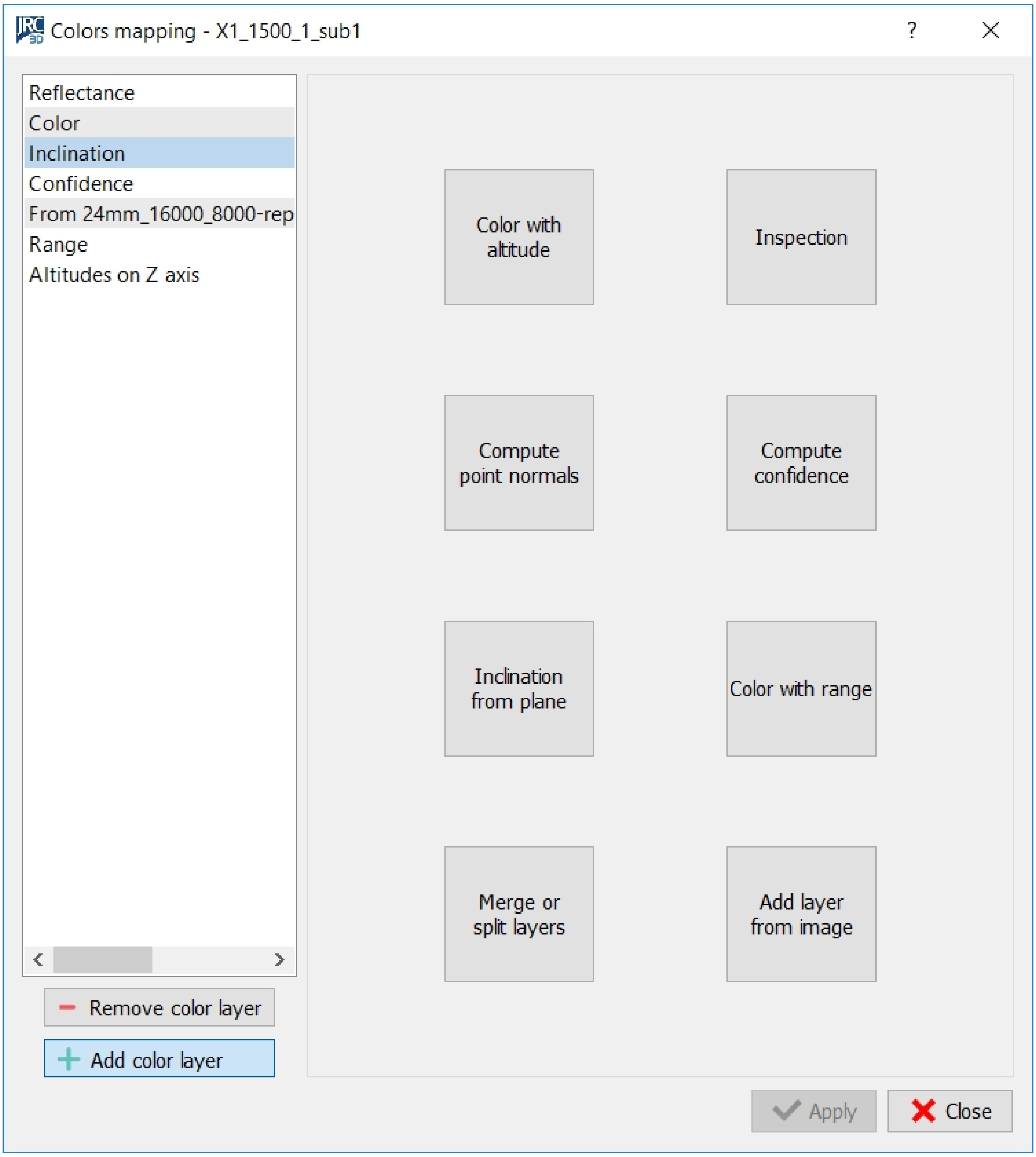
If selected color with default (Gray color determined by reflectance value) is applied else the colors map is applied.

Histogram tab



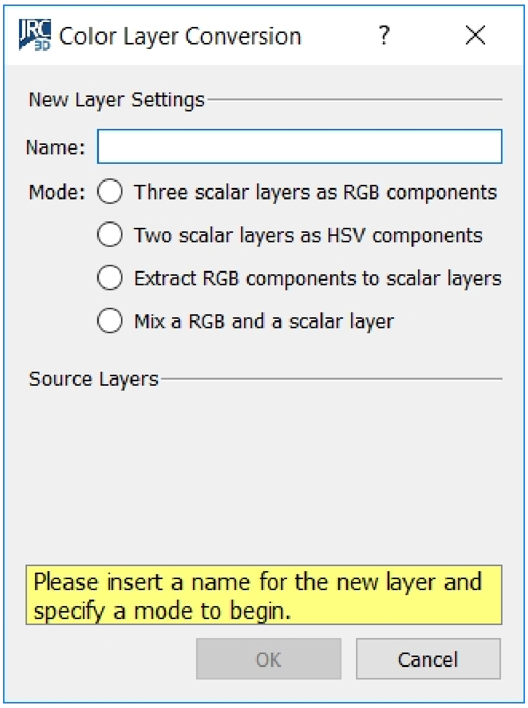
A colored histogram shows a statistical distribution with colors chosen by user. In abscissa, the histogram information about current measure (each color identifies a local radius of the measure) and in ordinate it shows a measure of how many occurrences of points are mapped to a color. The mean value is also shown.

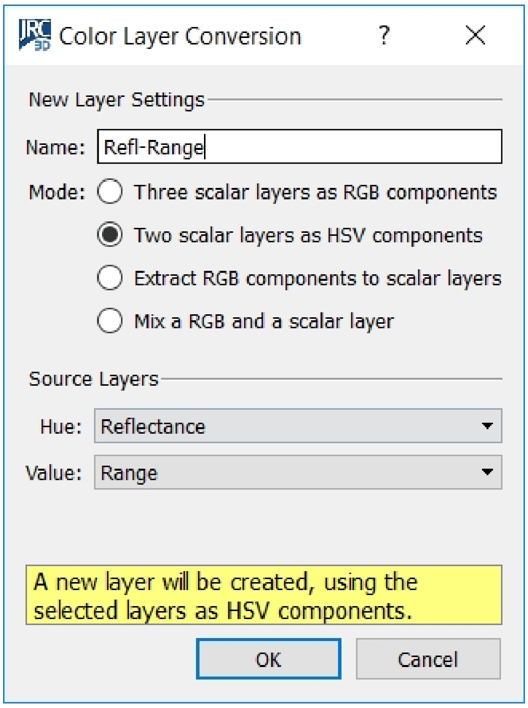
Add color layer tab



User can add color layers to a cloud via functions like *Inspection* or *Color with altitudes* by clicking buttons in *Add color layer* tab.
Buttons to add layers are:

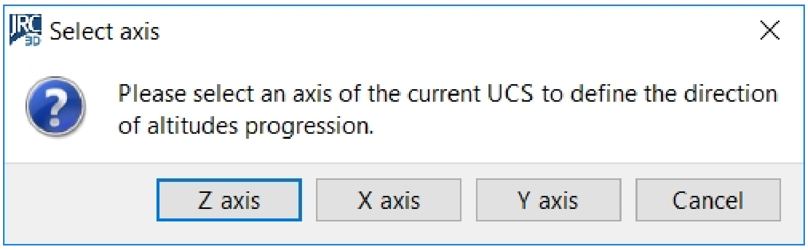
- *Color with altitudes.* that insert a layer to color distances along an axis direction.
- *Inspection* that insert a layer to color distances between point cloud and a reference model (e.g. Mesh) .
- *Compute point normals* that insert a layer to colors normals of points coherent to a plane direction.
- *Compute confidence* that insert a layer to color confidence amplitudes.
- *Inclination from plane* that insert a layer to color inclination angles.
- Color with range that insert a layer to color distance from the observation point.
- Add layer from image that apply an image on the point cloud (e.g. a 2Dview after a color editing)
- Merge or split layers that combine RGB and scalar layers to obtain a customized new layer of color.





Color with altitudes

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.



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.

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 prespective projector;

- **Orthographic:** creates an orthographic projector;
- **Spherical:** creates a spherical projector;
- **Cylindrical:** creates a cylindrical projector.
-

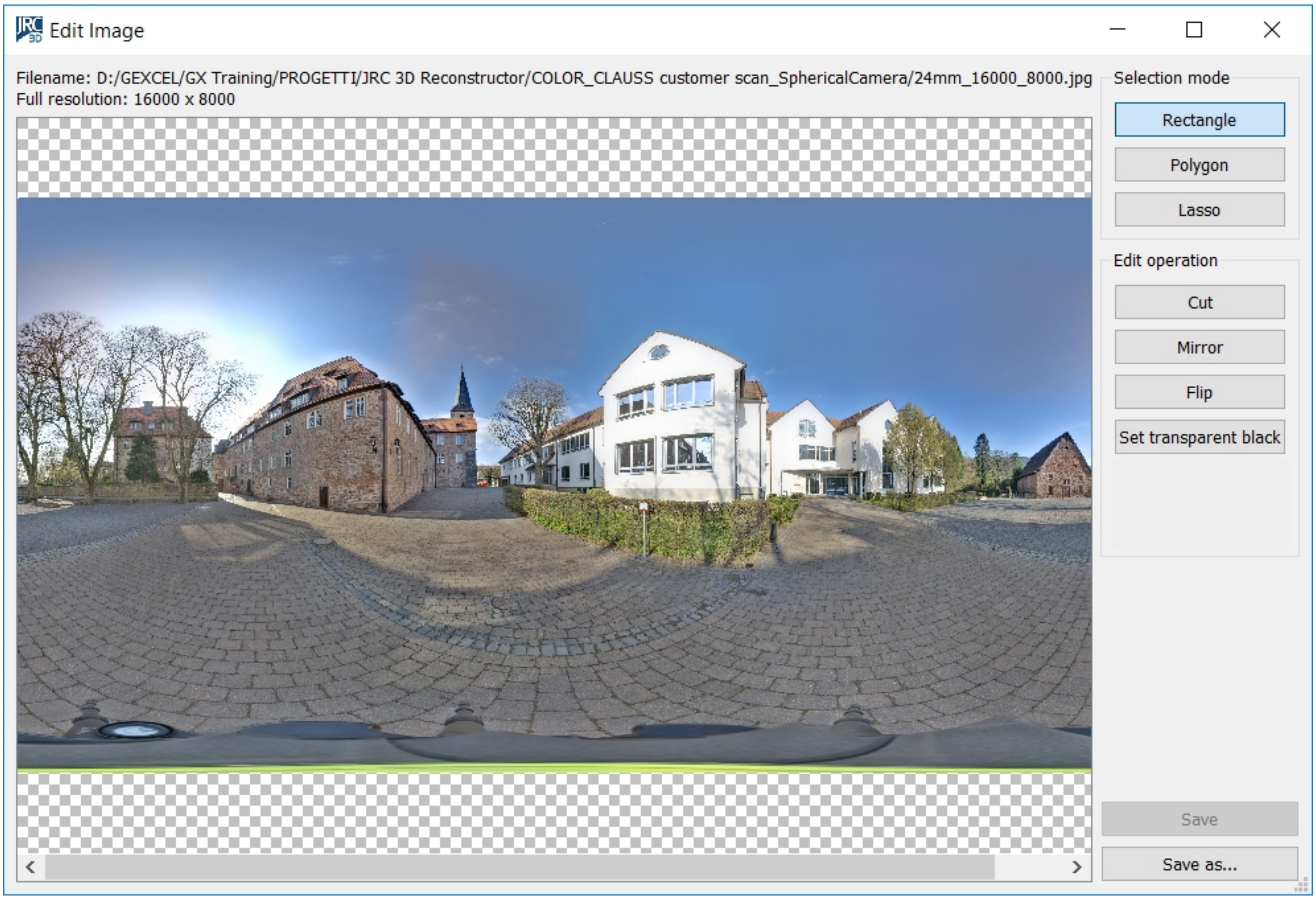
Note that a valid image is needed in order to create a projector.

See also Cameras.

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.

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.

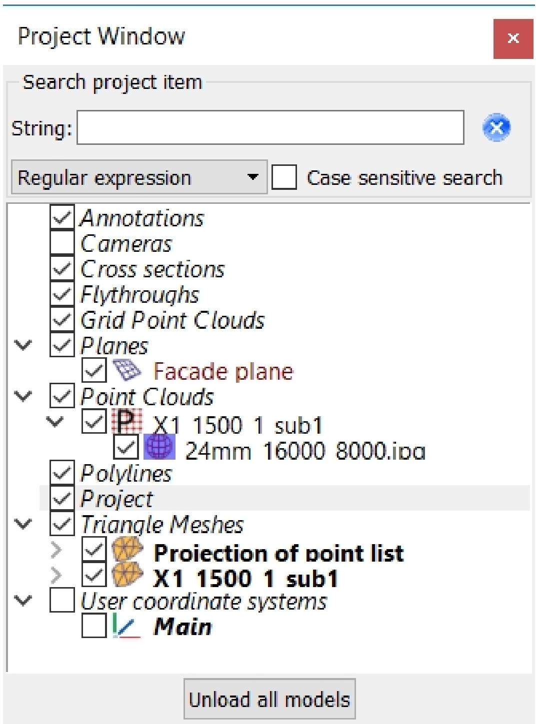
See also projectors.

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



This window provides a *tree view* on the current project items.

The Project window lists the items loaded in the project, organized in a tree structure, where each data type is grouped in a separate folder.
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.

Each item can be manipulated by its context menu (right mouse click) which displays its commands. Each item has properties that can be viewed and edited in the Properties window when the item is highlighted.

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. 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.

To remove an item from the current project, in the Project window select the item and press DEL 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.

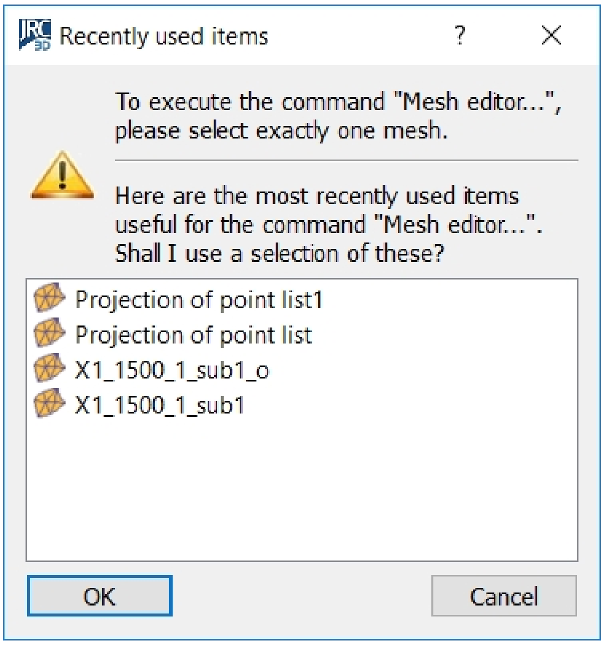
Multi-selection of items is possible by using Shift and Ctrl keys. Then it's possible to

- delete the selection with DEL 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 also dockable windows, and project items.

Recently used items dialog



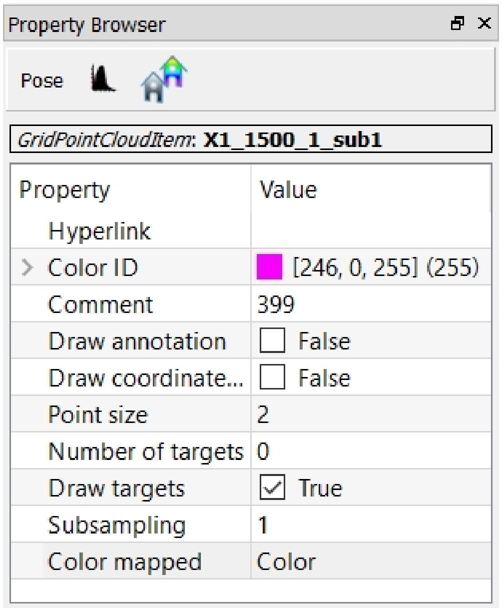
This dialog allows you to quickly select for a certain operation some of the project items you used most recently.

Many important JRC 3D 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 JRC 3D Reconstructor®'s GUI. It allows to read and edit the *properties* of the current selected item(s) in the project window.

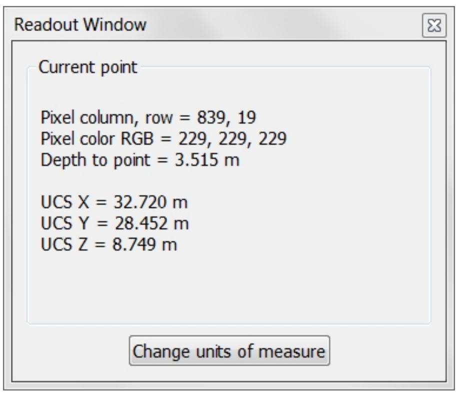
In JRC 3D Reconstructor® each project item has *properties*. There are *general* properties, owned by all project items, and there are *specific* properties that are peculiar to each type of project items.

General properties

These properties are common to all JRC 3D Reconstructor®'s 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.

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. It's accessible by right clicking on the Top toolbar.

The readout window displays information about the 3D point hovered by the mouse at a certain instant:

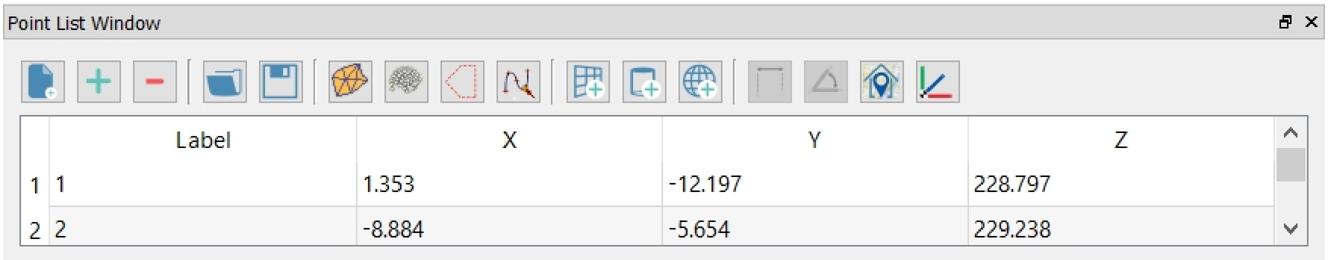
- *Pixel column, row:* refers to the mouse position inside the 3D window, in pixels (0, 0 is the top-left corner)
- *Pixel color RGB*: the three components of the point's *RGB* are shown
- *Depht to point*: distance beetwen selected point and current view point is displayed

- UCS X,Y and Z the coordinates of the corresponding 3D point, expressed in the current UCS, are displayed.

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.

Point List Window



The *point list window* is one of JRC 3D Reconstructor®r's dockable windows. It is situated by default on the bottom left of the GUI. 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 of your 3D models. The selected points appear annotated in the 3D scene and listed in the point list window.

The toolbar of the point list window

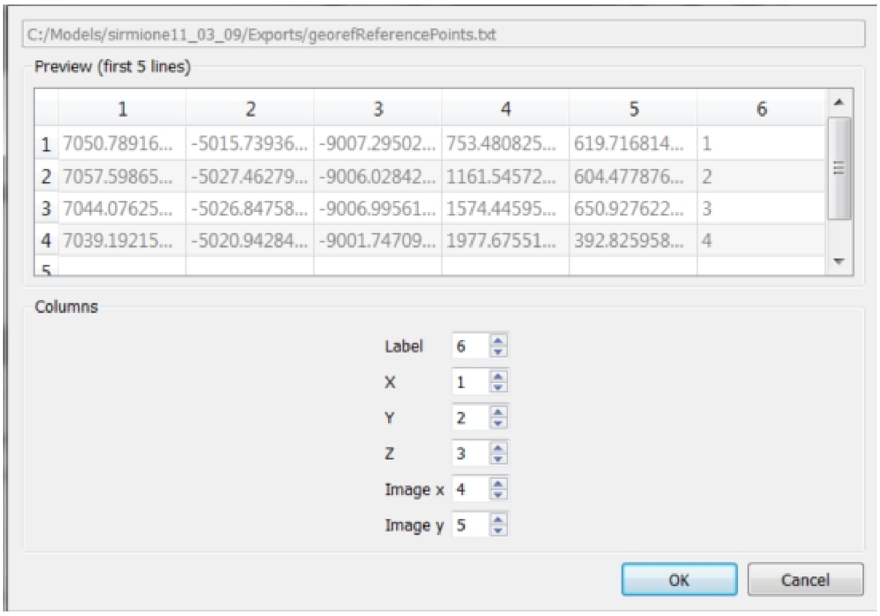
Once you have collected some points, you can perform several operations on them. These are accessible through the point list window toolbar. Below, the toolbar buttons are described left to right:

- **Clear**: removes all the rows in the list.
- **Add point**: adds an empty row in the list, to input manually 3D coordinates.
- **Remove point**: removes the currently selected row in the list. With Shift and Ctrl keys multiple rows can be selected, and deleted with the DEL key.
- **Load point list**: loads a list of points by parsing them from a text file via the parse point list tool.
- **Save point list**: saves the point list to a text file, writing one row per point in the format label, x, y, z.
- **Create triangle mesh**: this options creates a triangle mesh using as vertices the points listed in the window. 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.
- **Make point cloud**: the current list of points is saved as an Unstructured Point Cloud and added to the current project, so it can be exported to other formats.
- **Create polyline**: this option will store all the points in a polygon and save it as a polyline in the project. JRC 3D Reconstructor® asks you whether you want to create an open or close polygon.
- **Fit trajectory**: creates a video trajectory that runs throughout the listed points. The trajectory is created by interpolating a spline through the points. You can select the up vector of the video camera that will run through the points. The created trajectory is saved in the project under the name “flythrough from point list”.
- **Fit plane, cylinder, sphere**: if at least 3 points are defined a fit can be computed and added to the project.
- **Measure distance**: if 2 rows are selected, the distance of the 3D points is computed, even if they are very far apart.
- **Measure angle**: if 3 rows are selected, the angle of the 2nd point relative to the 1st and 3rd is

computed, even if they are very far apart.

- **Geo-referencing from point list**: starts a registration procedure that registers the listed points against other points that you consider to be reference points. This is mainly useful for geo-referentiation. This procedure has the effect of moving all your project items, including annotations, according to the resulting registration transform.
- **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.

Parse point list dialog

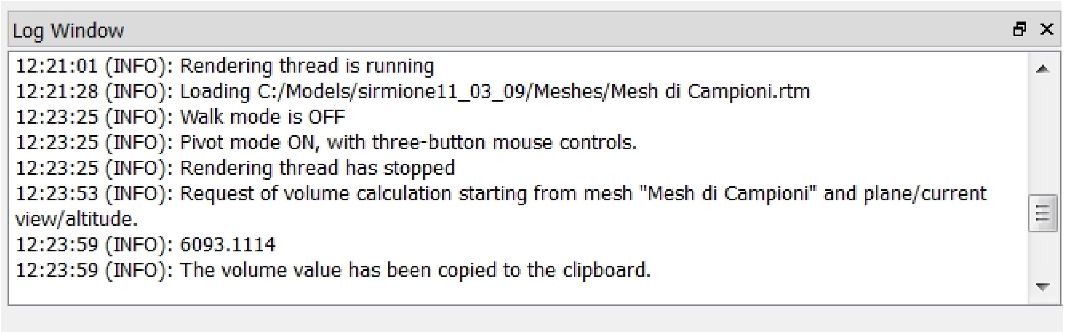


This dialog implements a general-purpose tool for parsing a text file with a list of points.

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 three columns contain the 3D coordinates of the point, column four and five contain the X and Y coordinates of the point in the range image (or in any image), and the sixth column contains the label of the points. Using the spin boxes in the bottom half of the dialog, you can customize the role of each column of the file.

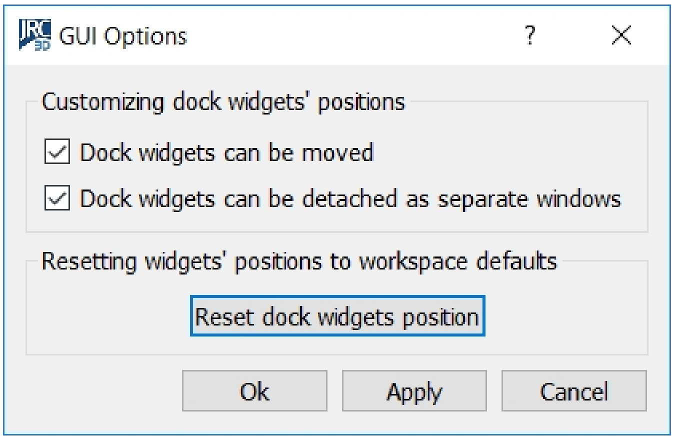
This tool is used to parse point files in the following procedures and windows: Camera calibration, importing a polyline from .txt format, Geo-referentiation, Point list registration, and pre-registration.

Log Window



Button clicks, status of algorithms, errors and warnings, result of procedures... everything that happens inside JRC 3D 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.
The log window is by default docked in the bottom right area of the GUI.

GUI Options dialog



This dialog allows the user to change some GUI settings.
The first two checkboxes control how freely the dockable widgets can be moved around and detached.
There are also two buttons to reset widgets' positions to a default. These defaults are decided by the current workspace configuration, internally known by JRC 3D Reconstructor®.

Help

Through this menu you can access JRC 3D Reconstructor®'s Online Help system. Furthermore, you can also access diagnostic information, useful when asking for support. Moreover, you can enter JRC 3D Reconstructor®'s license manger, to browse the license features activated and activate new ones.

- Online help
- Video Tutorials
- About plugins
- About JRC 3D Reconstructor®

Online help

This menu voice opens the present online help (which can be invoked also by pressing **F1**).
You can also open the contextual online help about the current window by just pressing F1.

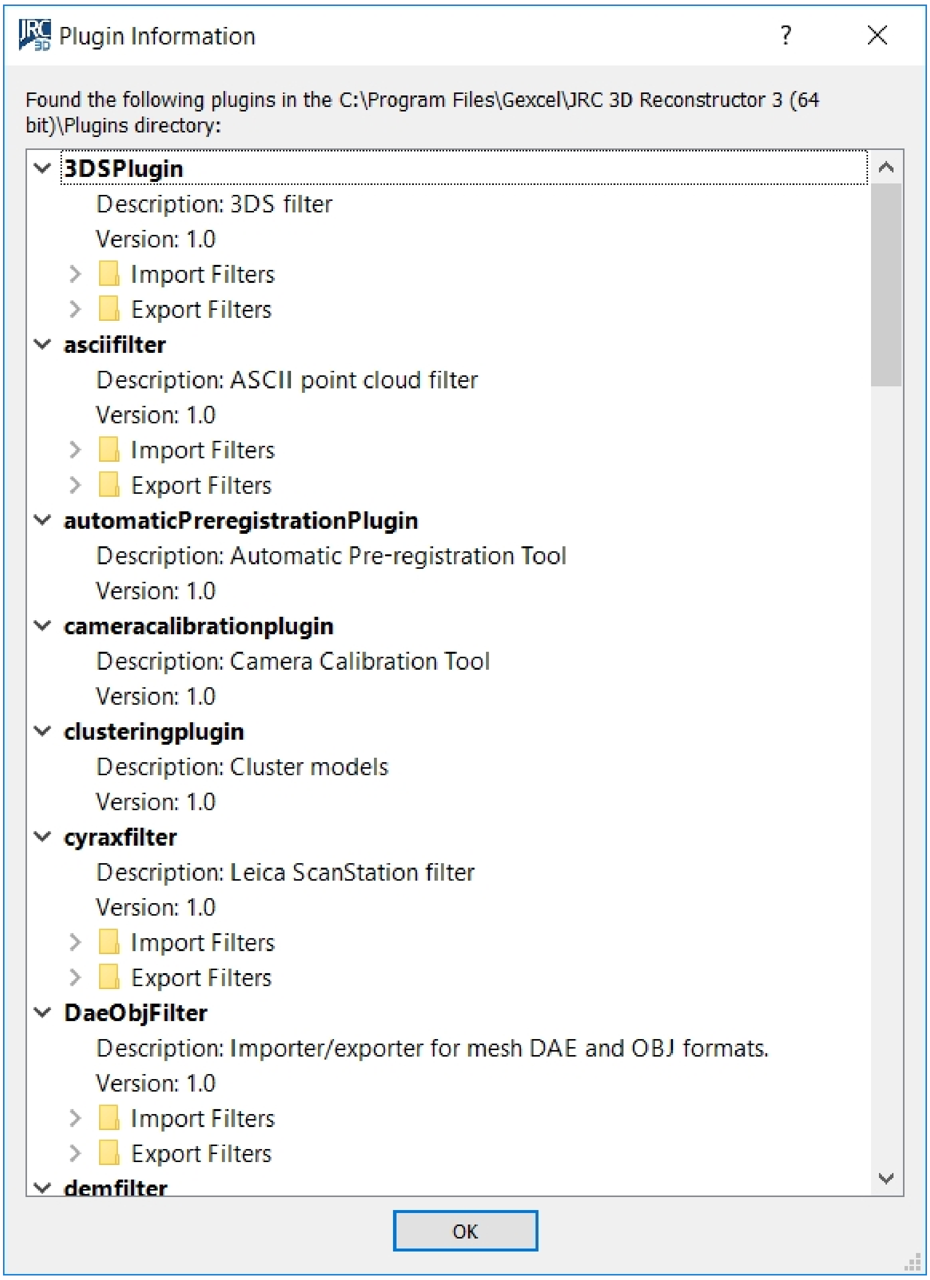
Video Tutorials

Please click on the following links to browse a list of JRC 3D Reconstructor®'s video tutorials on the web.

> Watch YouTube videos
> Watch YouKu videos

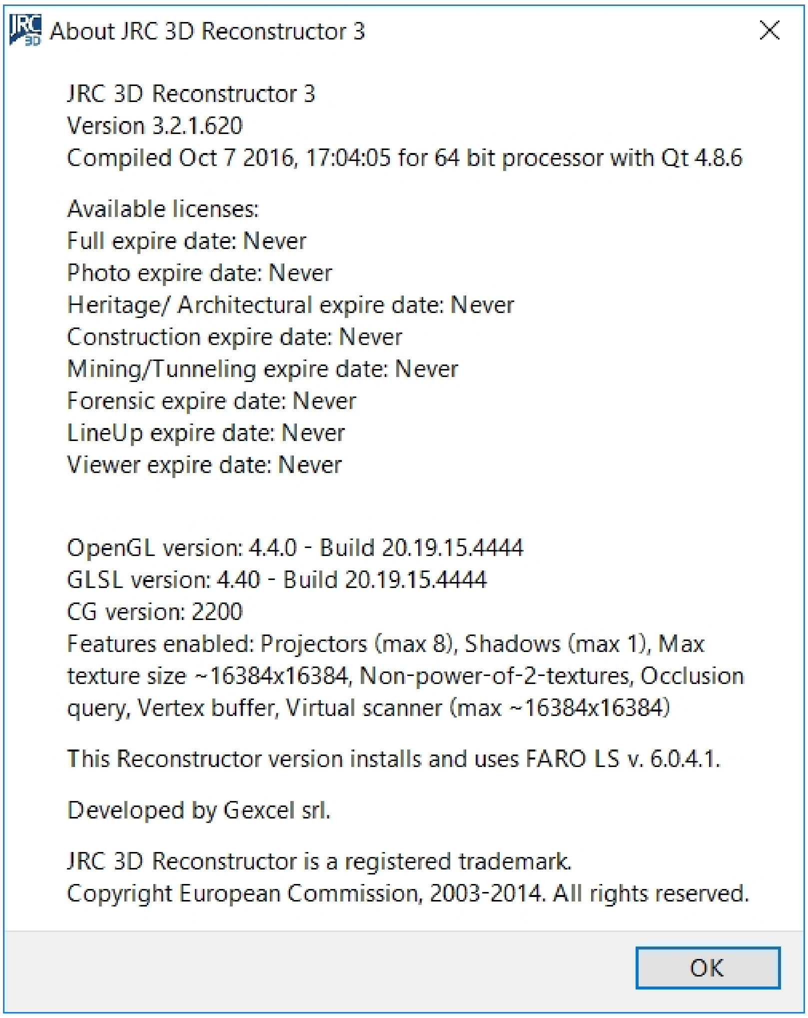
You can also find other tutorials on the relative pages of software's description.

About plugins



The menu option *About plugins* shows you a list of plugins. This is useful to know which import/export formats are currently available in JRC 3D Reconstructor®, and to know if a particular algorithm (e.g. LM-ICP bundle adjustment) is installed or not. When asking for support, you can be requested to send a screenshot of this window.

About JRC 3D Reconstructor®

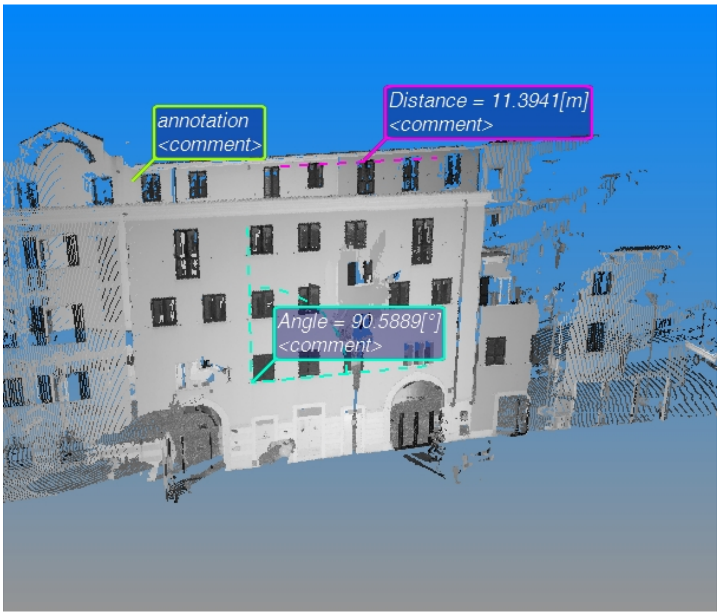


This menu option opens the *About JRC 3D Reconstructor® 3* window. This window displays JRC 3D Reconstructor®'s complete version number and other diagnostic information. When asking for support from Gexcel s.r.l, you may be asked to produce a screenshot of this dialog, or to report some information from this window.

Glossary

In this section you can find all the main definitions that you can encounter during the usage of JRC 3D Reconstructor®.

Annotations



JRC 3D Reconstructor® allows you to place annotations of your 3D models in the scene. These annotations may appear as items in your project. Annotations can be of three types:

- *Point annotations*
- *Distances*
- *Angles*

You can create an annotation via the menu options in the *Output->Measures & Notes -> Annotation* menu or the corresponding buttons in the top toolbar. To create a punctual annotation, select the "annotation" button from the aforementioned menu and double-click on any point of your 3D scene.

Distance annotations are used to measure the distance between two points in the 3D scene. Select *Output->Measures & Notes ->Distance*, then press the left mouse button on the first point of the segment to measure, move the mouse and release the left button on the end point of the segment. JRC 3D Reconstructor® shows you the distance measured and asks you whether you want to store it into the project.
Selecting the *Output->Measures & Notes ->Angle* button, you have also the option to measure angles. You define the angle you want to measure by left-clicking on three points of your 3D scene.
In the properties of any annotation, you can change its color ID, edit the displayed comment, define a hyperlink for an annotation (interesting feature to add meta-information to your 3D scene from e.g. the web) and choose whether to draw the annotation or not.

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 adjust pose or the pose dialog.

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.

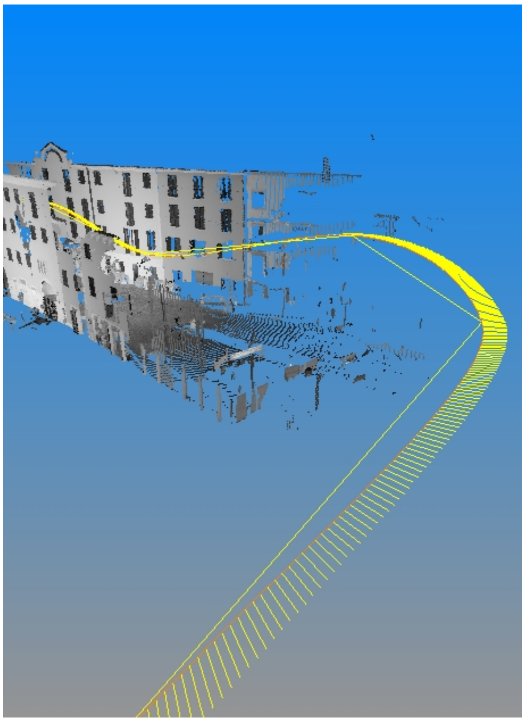
From the contextual menu of any camera, you can activate also the following commands:

- **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 ortho)
- **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 "Maximun incident angle"
    b) have a frustum that intersects the frustum of the selected camera
    The projectors/cameras are so checked in the project window
  - **Select models in this frustum** finds (and checks) the meshes intersecting the selected camera (useful for texture mapping)
- **Elevation&Plan->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.

Flythroughs (video trajectories)



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 via the point list window. 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 in total freedom.

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. These three options are also available through the top menu, via *Outputs->Video record*.

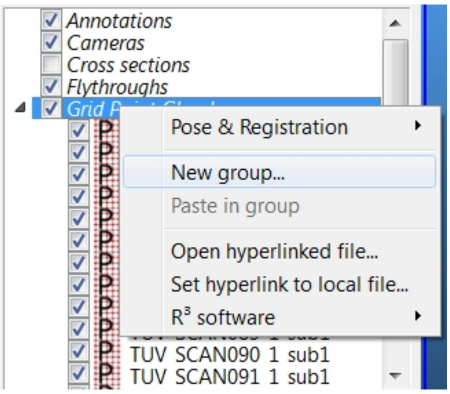
From the flythrough contextual menu you can access the submenu *Tunnel survey* with two functions designed for flythroughs:

- *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, JRC 3D 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. JRC 3D 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, JRC 3D Reconstructor® asks you the spacing between different cross sections along the trajectory. When the spacing is set, JRC 3D 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 contextual menu.



To *create* a group you only have to open the contextual menu of a data type folder by right-clicking on it and select the ***New group...*** command.

To populate a group you have:

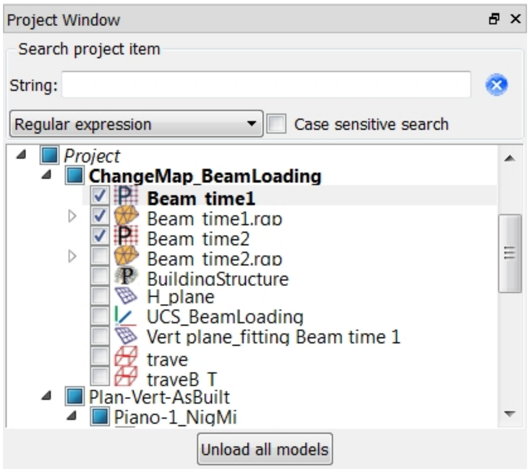
- to move a selection of items from a previous position (to a new one) with ***Move*** command from the contextual menu of the selection
- to paste the selection of items in the new group, with ***Paste in group*** command from the contextual menu of the new group.

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 (for example
- *Children move together*: if a single scan is moved in the 3D space, all the other scans move together, as a rigid system

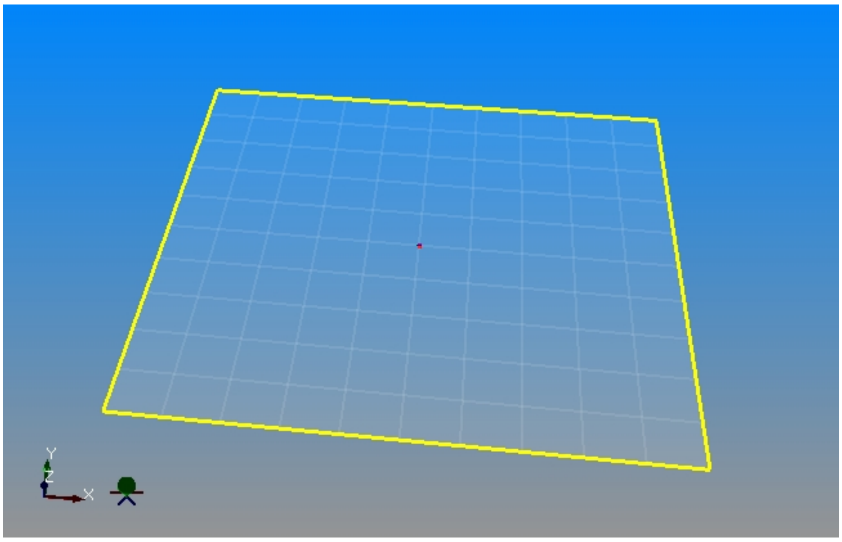
These roles are used to make a scans registration through groups.

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 contextual menu of the object is the way.

Planes



JRC 3D 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 JRC 3D Reconstructor®:

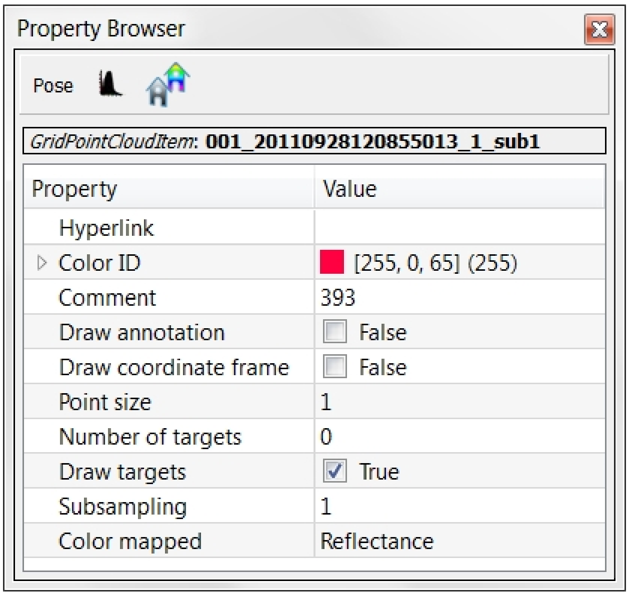
- ***Structured*** point clouds (also called *grid point clouds,* or *range images*, or *scans*): scan-based clouds, "fixed-origin" or tripod scans. They include row/column info and the origin scan's position.
- ***Unstructured*** point clouds

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 Options

A Grid 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).

2D pixels of the Grid		3D points			RGB value		
1	452	-0.026171	-3.7225	-7.1416	104	104	104
1	455	-0.026171	-3.7225	-7.1416	171	171	171
1	459	0.16056	-3.0731	-1.6781	70	70	70
1	460	0.16513	-3.0732	-1.6781	129	129	129
1	461	0.16969	-3.0733	-1.6782	174	174	174
1	462	0.17425	-3.0734	-1.6786	174	174	174
1	463	0.17882	-3.0735	-1.6787	168	168	168
1	464	0.18337	-3.0737	-1.6792	171	171	171
1	465	0.18793	-3.0738	-1.6795	168	168	168
1	466	0.19246	-3.074	-1.6803	154	154	154

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).

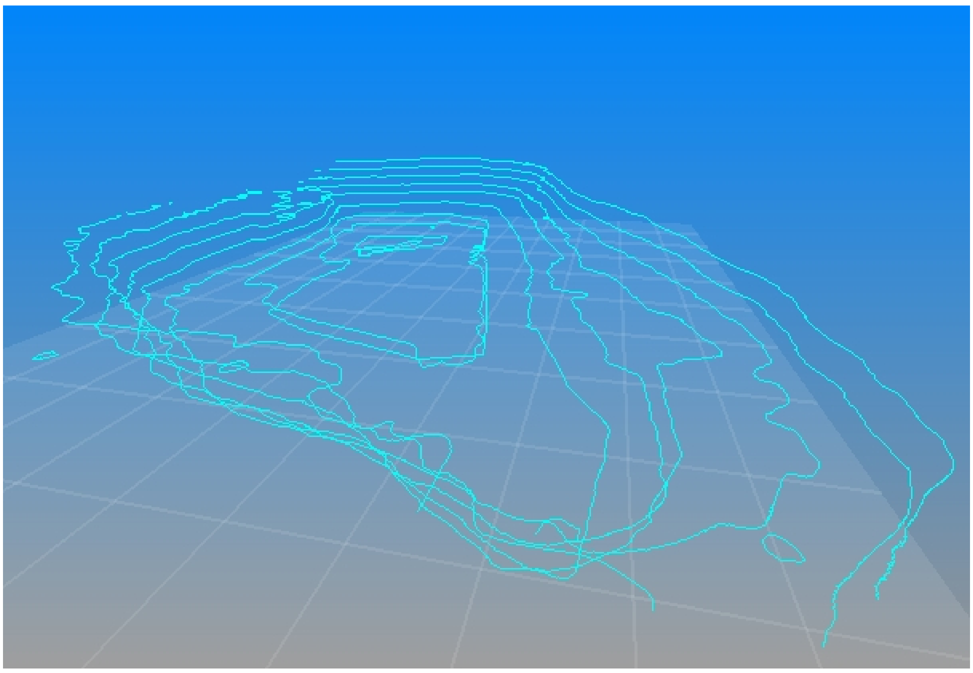
For point colors, a special 3bc (c stands for compressed dynamic range) file is created for rendering only, for instance after histogram equalization.

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"

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 *Tools->Meshing->Get mesh borders as polylines*, belonging to the contextual menu of the triangle meshes.
- From the Mesh ridges&valleys tool.
- From the edge extractor of grid point clouds, available via *Tools->filtering->extract edges...*
-

The most important contextual 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.

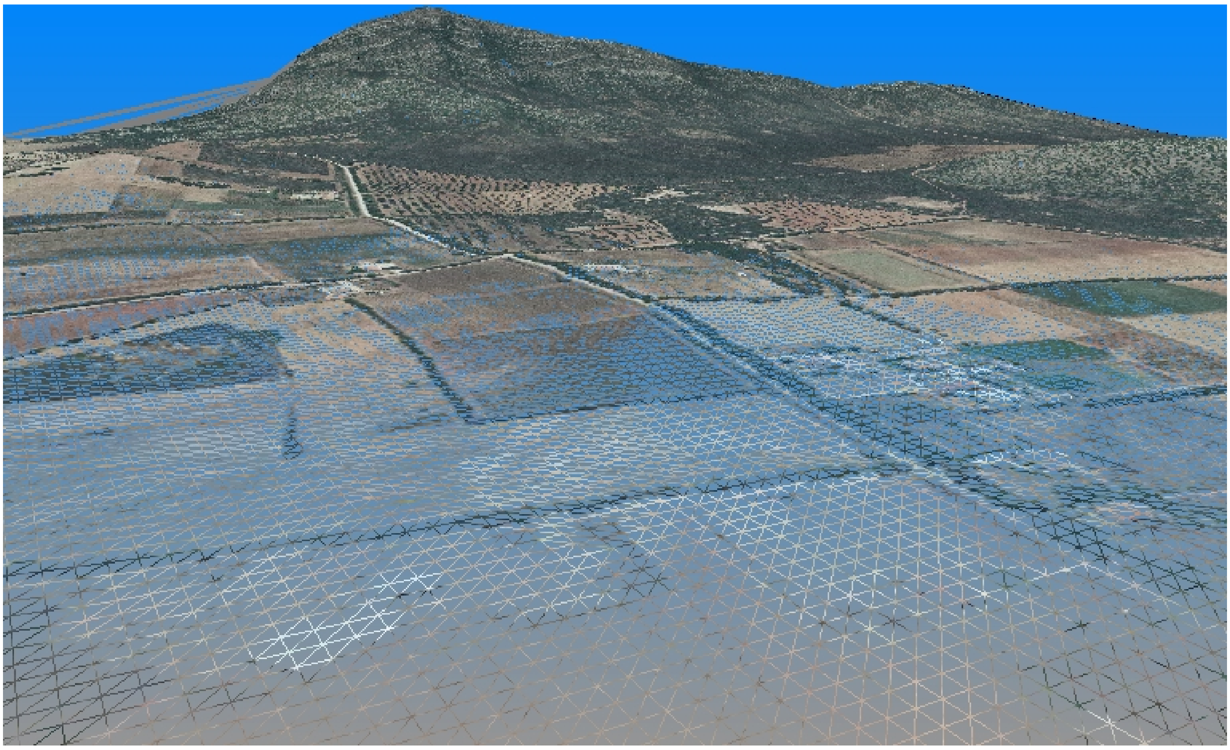
The specific commands are:
Load projector: to load (light on) the image on the 3D model.
Set projector image: to set the desired image to project
Edit projector image: edit the projector's image to set the transparent pixels in order to produce better blending among the projectors

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

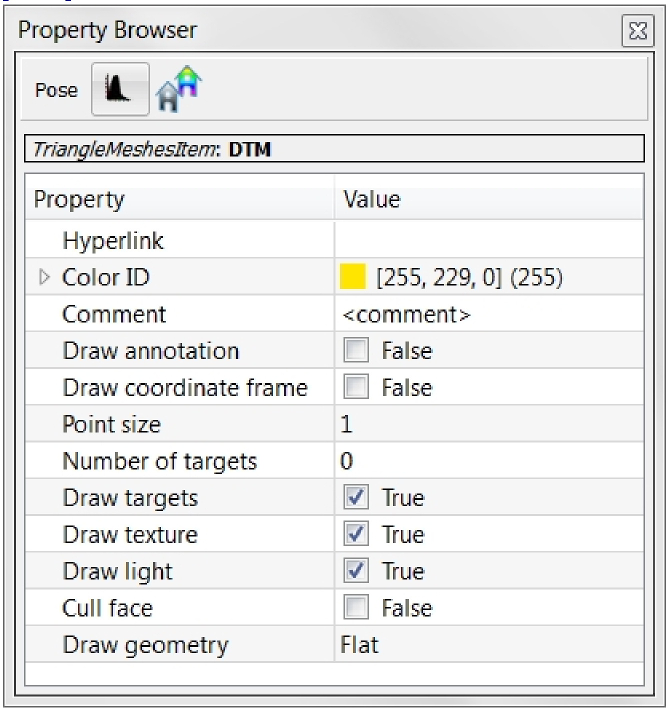


A triangle mesh is a 3D model represented by a set of triangles connected by common edges and common vertices. If a point cloud includes a set of points, a mesh includes more information because it stores how the points are connected.
A triangle mesh therefore defines a surface in the 3D space. The mesh's 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.
JRC 3D 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

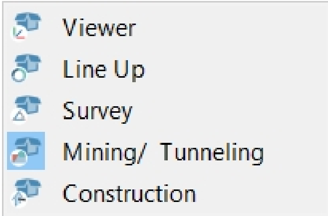
Data is stored in an XML description file + pure binary vector files subdivided in folders for each submesh. The file extension of the binary data specifies the format, where data.NT: N=num of components (1,2,3,4...), T=data type (byte, short, word, half, float, double). If a mesh is called myMesh, a 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.

Workspaces

A *workspace* is a particular configuration of JRC 3D Reconstructor®'s interface, aimed at satisfying a particular workflow.



Note: the access of each workspace is due to the available licenses.

See here for details.

User Coordinate System (UCS)

User coordinate systems are a central concept in JRC 3D 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).

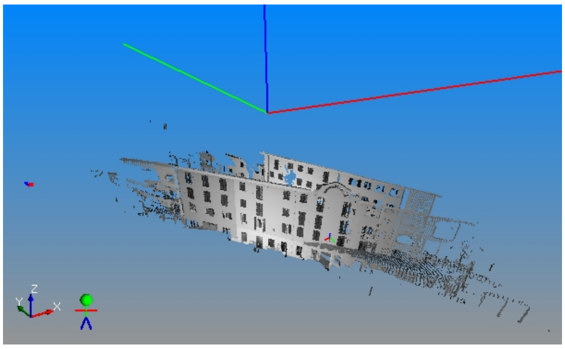


Figure 1

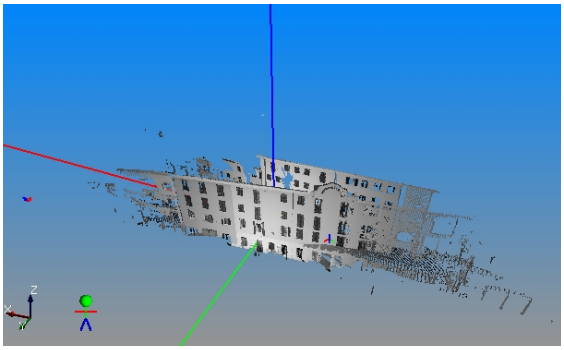


Figure 2

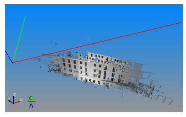


Figure3

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 JRC 3D 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 JRC 3D 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 JRC 3D 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.

Create UCS from this pose

The contextual 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 contextual 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).