Sterio Framework User Guide

1. Creating a project

After starting the application, select one of the two options in the upper menu (Fig. 1). In order to create a new project, click the "New" button, select a storage location and the project name. If you are opening an existing project, press the "Open" button and select the project file with the ".sterio" extension.

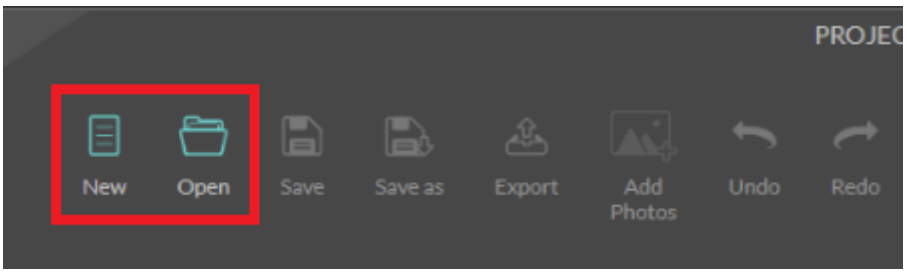

Fig. 1 Project creation/opening buttons

2. Adding photos to a project

After creating or opening an empty project, add photos. To do so, click the "Add Photos" button located in the main menu, in the "PROJECT" tab (Fig. 2).

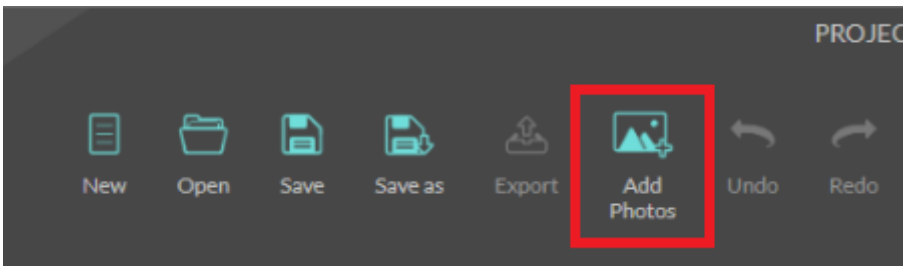

Fig. 2 Add photos button

After selecting this option, a panel with the same name will be displayed, which will be used to load photos to the project. If there is a need to load several versions of photos, for example photos with additional lighting or with a pattern from a projector, select their number from the drop-down menu at the top of the panel (Fig. 3). Each version is divided into 'left' and 'right' photos, the number of which must be the same (but the number of photos between versions may be different). If you upload photos to an already existing version, select the "To existing" option to the right from the version.

Fig. 3 Add photos panel

After selecting the number of versions and indicating the photos, the action is confirmed with the "SELECT" button, which will start loading the photos into the project. The loading process will be visualized by a progress bar (Fig. 4). The upload time depends on the number of photos and their version.

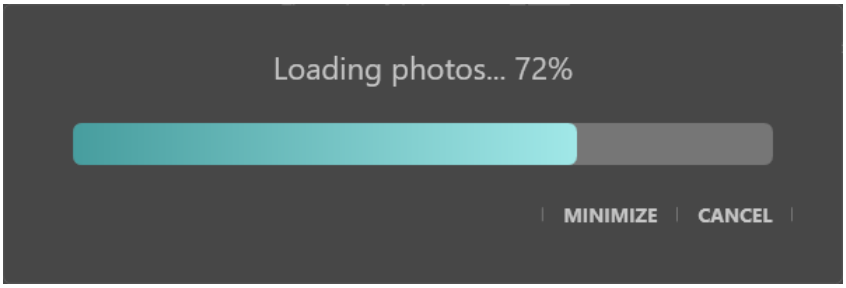

Fig. 4 Uploading photos to the project

After completing this process, you can view all the loaded photos in the lower section of the application: "PHOTO LIBRARY" (Fig. 5). Here you can see thumbnails of photos or double-click with the left mouse button on the photo data to see the full-size version in the default Windows application for displaying graphic files. By right-clicking on a photo or a group of photos, a context menu appears where you can delete unwanted photos, create a group or add selected photos to an existing group. To continue the reconstruction process, select the photos you want to use and select the "Create group" option.

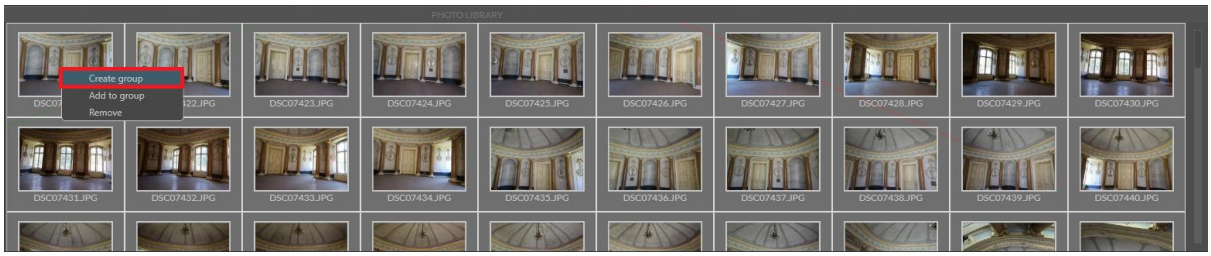

Fig. 5 Photo library

After creating a group, it will appear on the left side in the group and version tree, with a default colour and name (Fig. 6), which can be changed by right-clicking on the group or version and selecting the "Rename" option. In the same context menu the selected group can also be deleted. On the right side of the group name there is a control that allows you to change the colour assigned to the group by default (colour picker). On the left side there are

checkboxes with which you can hide photos of a given group or version in the "PHOTOS LIBRARY".

Fig. 6 Group before and after changes

3. Location of cameras and reconstruction of a sparse cloud

Once the photos have been loaded correctly and a group has been created, the next step in the reconstruction is to locate the cameras and generate a sparse point cloud. To start this process, click the "Build Sparse Cloud" button located in the main menu, in the "RECONSTRUCTION" tab.

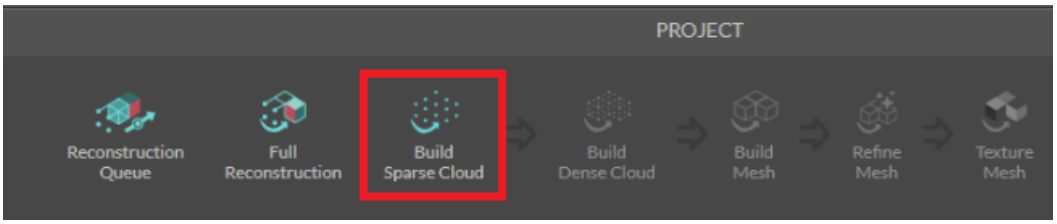

Fig. 7 Sparse cloud reconstruction button

Selecting this option will display the sparse point cloud reconstruction panel (Fig. 8). At the top of the panel, select the group and version to be reconstructed. Then customize the individual reconstruction options or select the default configuration values that will automatically change everything. The next step is to choose the camera model that was used to take the pictures in order to calculate the focal length of the camera in pixels which will be used to remove the distortions. The remaining options you can adjust are: Describer method, Distant ratio and Algorithm.

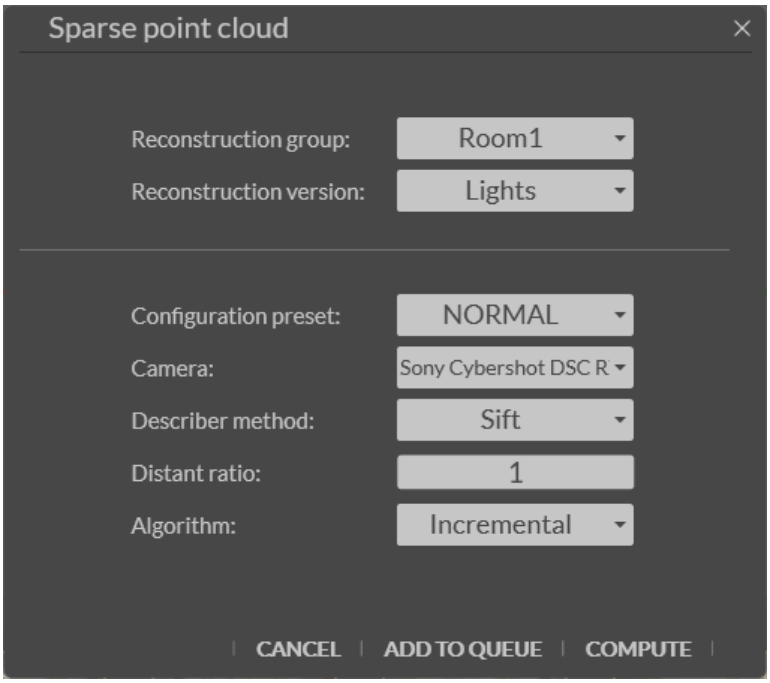

Fig. 8 Sparse cloud reconstruction panel

Once all options have been customized, click the "COMPUTE" button to start the process of locating the cameras and reconstructing a sparse point cloud. The progress of the process will be visualized in the panel with the progress bar (Fig. 9). The duration of this step depends on the number of photos, their resolution and the selected options.

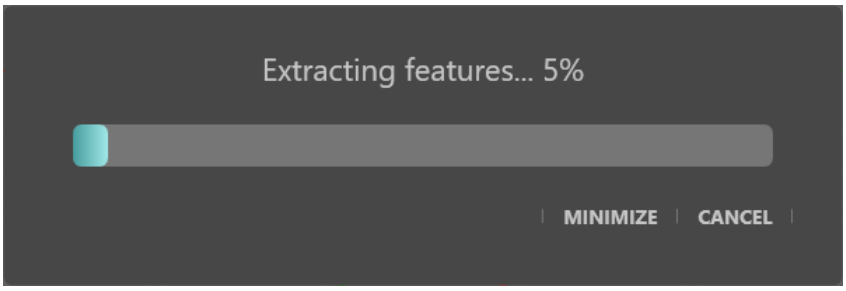

Fig. 9 Sparse cloud reconstruction

After the successful reconstruction of this stage, in the "PHOTO LIBRARY" section at the bottom of the application you can see which photos have been accepted (i.e. common points between them have been found) and which have been rejected. In both cases, the corresponding markers are displayed in the upper right corner of the thumbnail (Fig. 10).

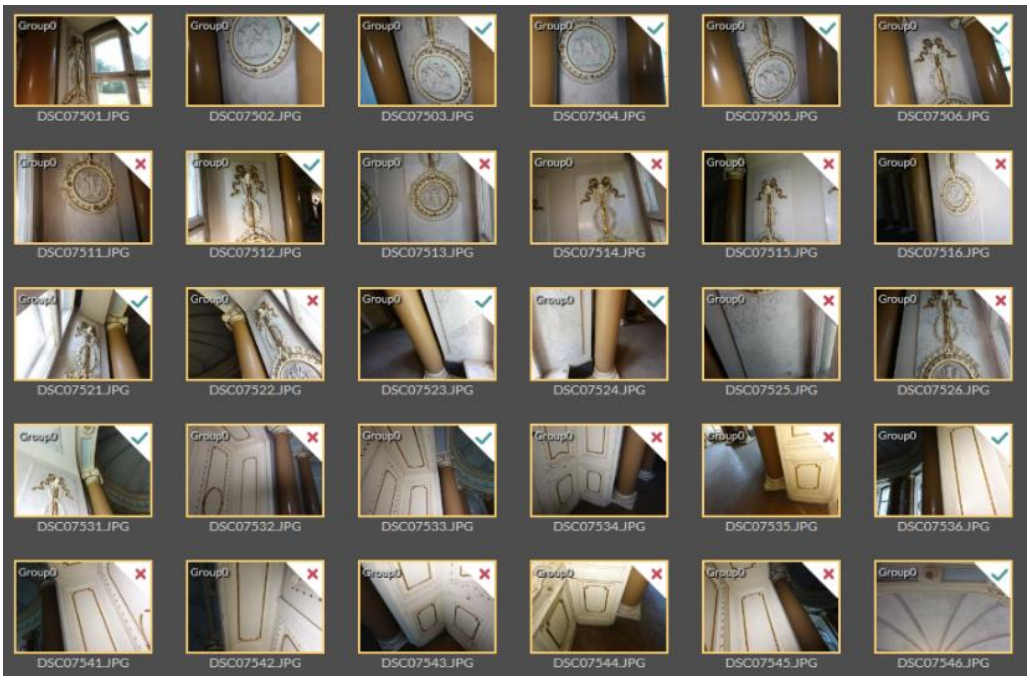

Fig. 10 Accepted and rejected photos

The reconstructed sparse point cloud should also load automatically (Fig. 11). It is only an illustration; any changes made to it will not be taken into account in the subsequent stages of reconstruction.

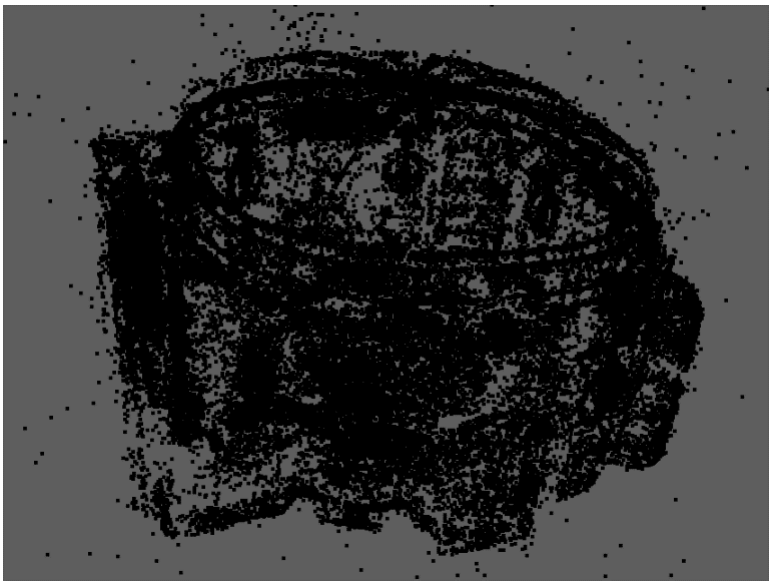

Fig. 11 Sparse point cloud

After selecting the "Show Cameras" button (Fig. 12), the locations and orientations of the cameras for the selected photos will be displayed (Fig. 13).

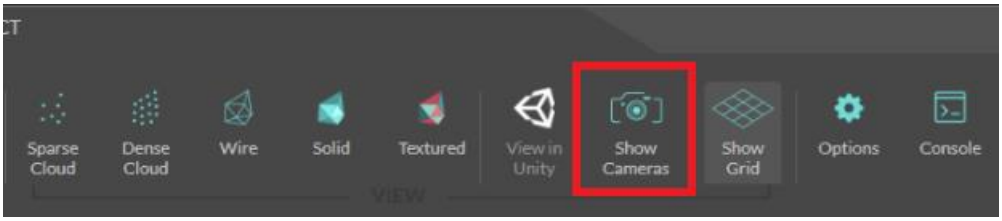

Fig. 12 Button for loading localized cameras

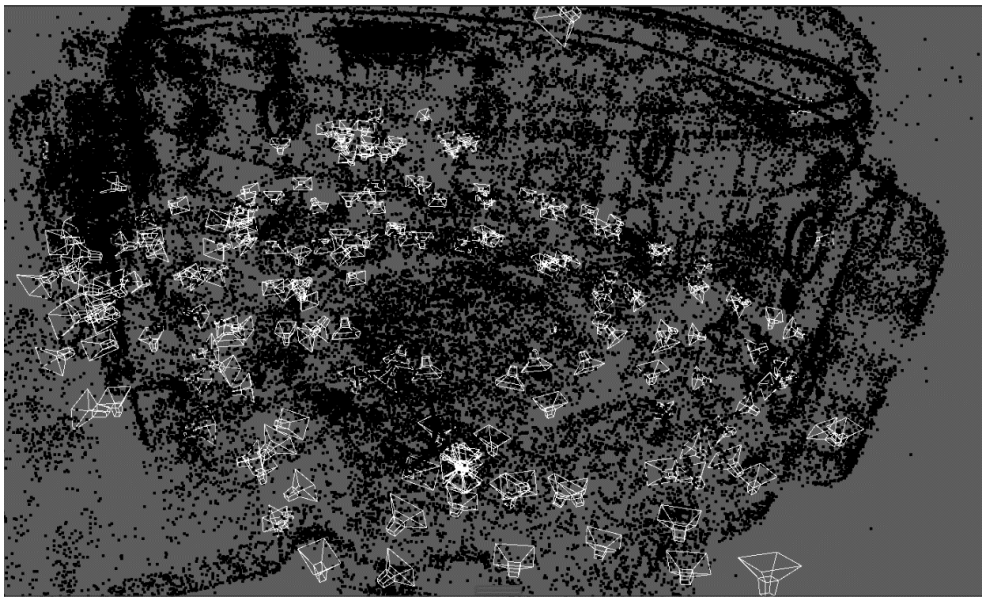

Fig. 13 Visualization of localized cameras

4. Reconstruction of a dense point cloud

After correct reconstruction of a sparse point cloud, the next step is the reconstruction of a dense point cloud. To start this process, click the "Build Dense Cloud" button located in the main menu, in the "RECONSTRUCTION" tab (Fig. 14).

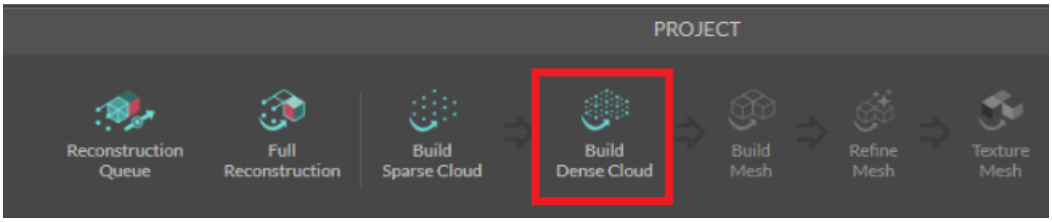

Fig. 14 Dense cloud reconstruction button

When this option is selected, the dense point cloud reconstruction panel will be displayed (Fig. 15). At the top of the panel, select the group and version to be reconstructed. Then adjust the individual reconstruction options or select one of the available presets that automatically change everything. Next, select the number of views used to calculate the depth map, the number of views fusion and whether colours and normals are to be estimated.

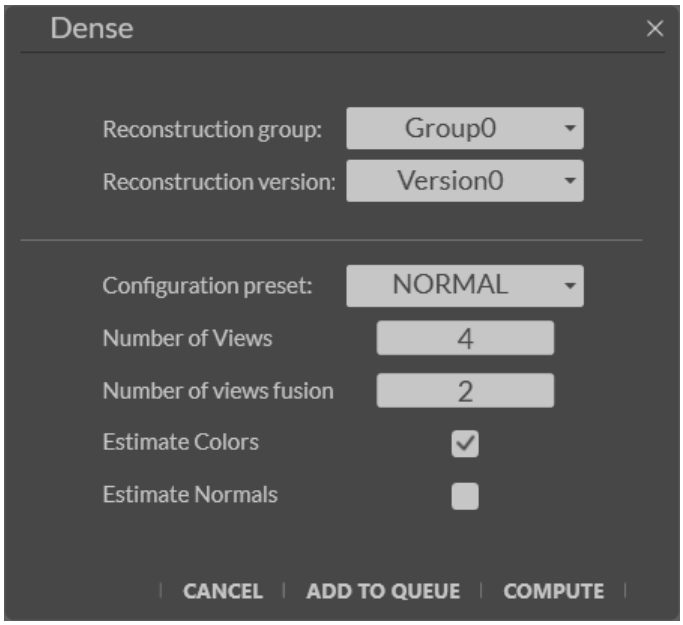

Fig. 15 Dense cloud reconstruction panel

Once all options have been adjusted, click the "COMPUTE" button to start the process of reconstructing the dense point cloud. The progress of the process will be visualized in the panel with the progress bar. The duration of this step depends on the number of photos, their resolution and the selected options.

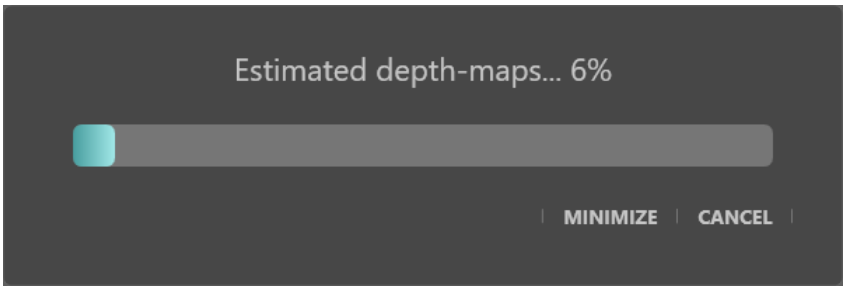

Fig. 16 Dense cloud reconstruction

Once the reconstruction of this stage is successfully completed, the reconstructed dense point cloud should automatically load and display (Fig. 17). At this stage, you can make changes such as removing points, flattening the surface, or moving and rotating the object. All changes will be taken into account in subsequent reconstruction processes.

Fig. 17 Dense point cloud

5. Model mesh reconstruction

After a correct reconstruction of a dense point cloud, the next step is to reconstruct the model mesh. To start this process, click the "Build Mesh" button located in the main menu, in the "RECONSTRUCTION" tab (Fig. 18).

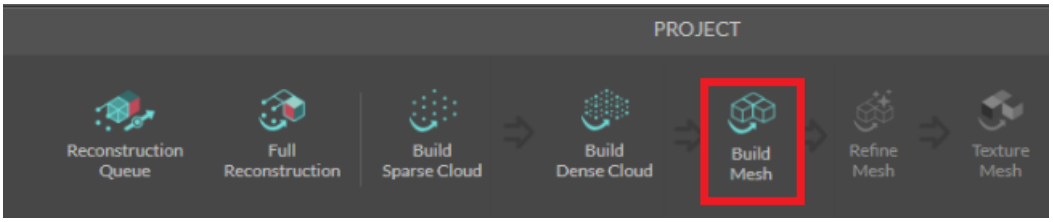

Fig. 18 Model mesh building button

When you select this option, the model mesh reconstruction panel will be displayed. At the top of the panel, select the group and version to be reconstructed. Then adjust the individual reconstruction options or select one of the presets which automatically change everything (Fig. 19). Next, select the minimum distance in pixels between two points to consider them as different in the process of triangulation ("Minimum Point Distance"), simplification factor in the range (0-1], which will be done on the reconstructed model ("Decimate"), incorrectness factor, which is used to remove triangles with too long edges or isolated components ("Remove Spurious"), whether to remove excessively protruding triangles ("Remove Spikes"), the factor used to try to close small holes in a reconstructed model ("Close Holes") and the number of iterations for smoothing the surface ("Smooth").

Fig. 19 Model mesh building panel

Once all the options have been adjusted, click the "COMPUTE" button to start the process of reconstructing the model mesh. The progress of the process will be visualized in the panel with the progress bar. The duration of this stage depends on the number of points in the dense cloud, as well as the selected options (Fig. 20).

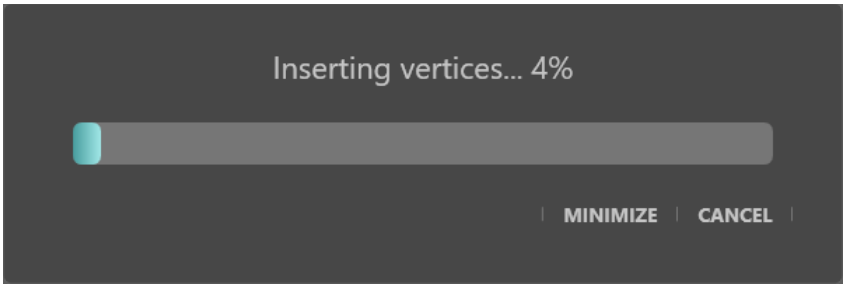

Fig. 20 Generating the model mesh

After successful reconstruction of this stage, the reconstructed mesh of the model should be loaded automatically (Fig. 21). Changes such as moving and rotating the object can be made at this stage. All changes will be taken into account in subsequent reconstruction processes.

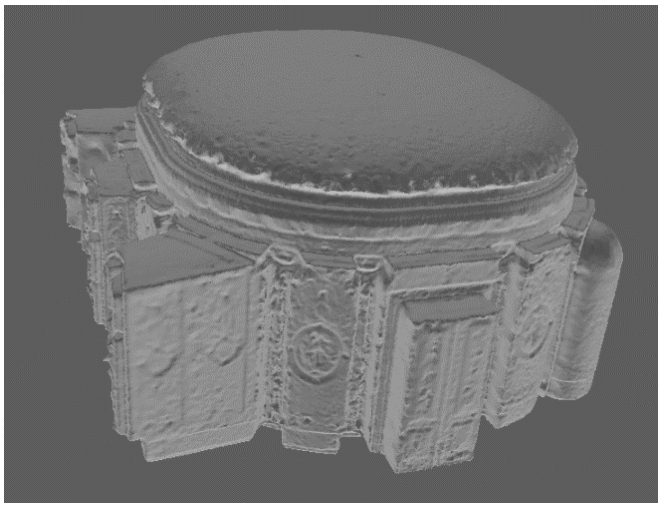

Fig. 21 Generated model

6. Model refining

After a correct reconstruction of the model mesh, the next step is to improve it. To start this process, click the "Refine Mesh" button in the main menu under the "RECONSTRUCTION" tab (Fig. 22).

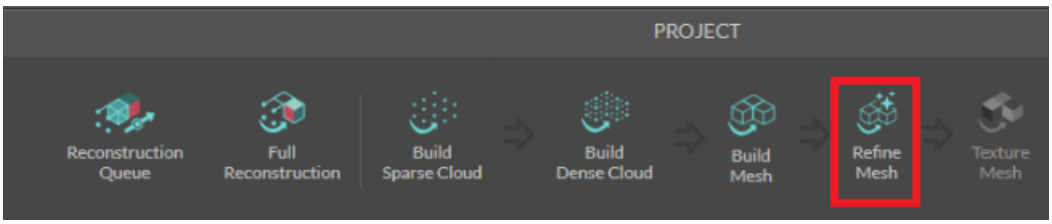

Fig. 22 Model refining button

When this option is selected, the model refining panel will be displayed (Fig. 23). At the top of the panel, select the group and version to be reconstructed. Then adjust the individual reconstruction options or select one of the available presets that automatically change everything. Next, select the maximum number of adjacent photos used for the reconstruction ("Max Views"), the simplification factor in the range (0-1] that will be used during the model refining ("Decimate"), the factor used for attempting to close small holes in the model ("Close Holes"), the threshold used to remove vertices on flat pieces of the model ("Planar Vertex Ratio") and whether the calculations are to be made using a graphics card ("Use GPU").

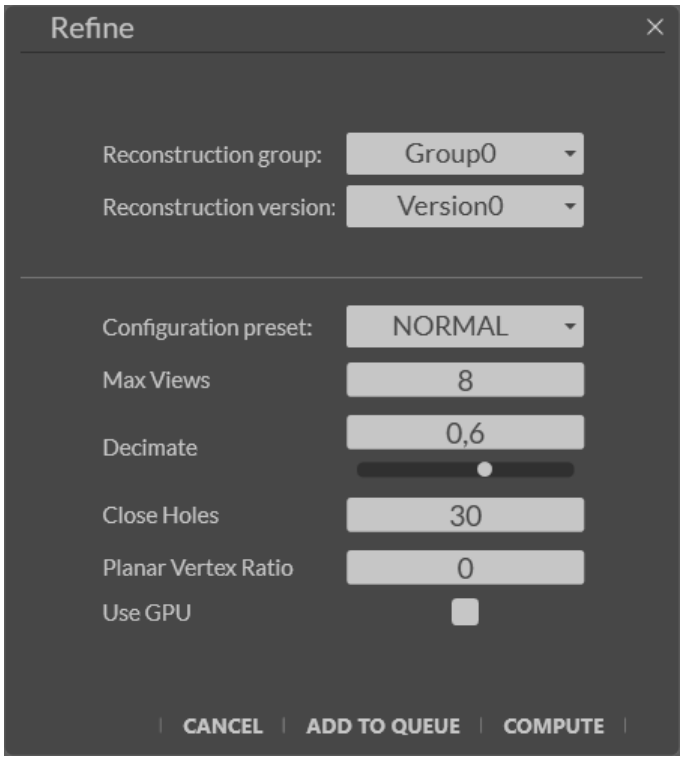

Fig. 23 Model refining panel

Once all the options have been adjusted, click the "COMPUTE" button to start the model refining process. The progress of the process will be shown in the panel with the progress bar (Fig. 24). The duration of this stage depends on the mesh density of the model and the selected options.

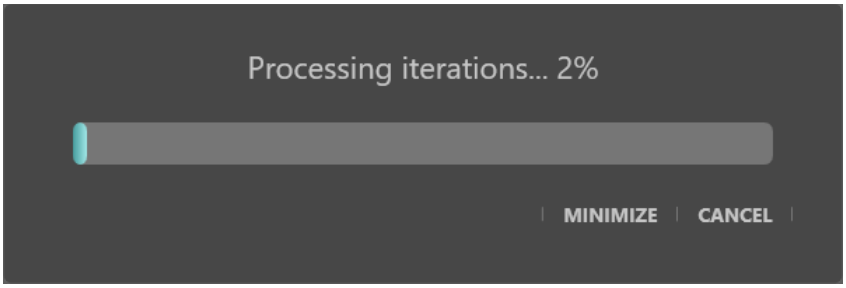

Fig. 24 Model refining

After successful reconstruction of this stage, the refined model should automatically load and display (Fig. 25). Changes such as moving and rotating the object can be made at this stage. All changes will be taken into account in subsequent reconstruction processes.

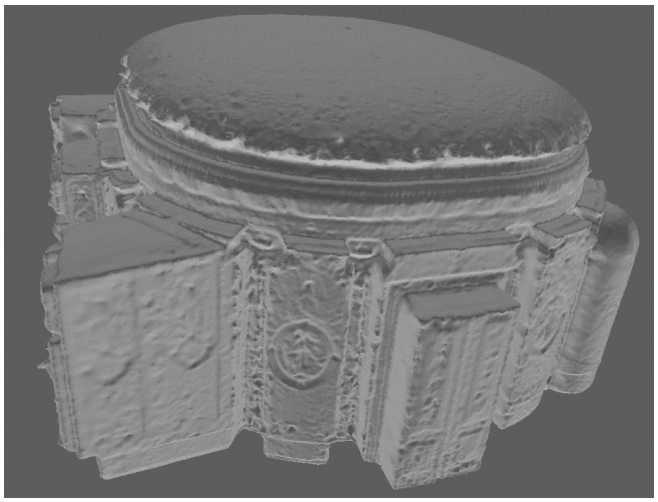

Fig. 25 Refined model

7. Model texturing

After the model has been properly refined, the next step is texturing the model. To start this process, click the "Texture Mesh" button in the main menu on the "RECONSTRUCTION" tab (Fig. 26).

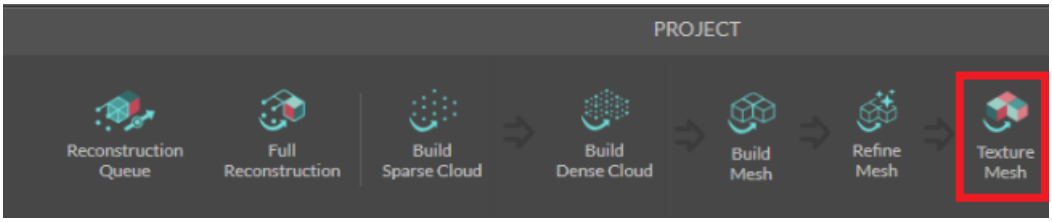

Fig. 26 Texture creation button

After selecting this option, the model texturing panel will be displayed (Fig. 27). At the top of the panel, select the group and version to be reconstructed. Then you can customize the reconstruction options or select one of the available presets. Then select the threshold used to find and remove outlier textures ("Outlier Threshold"), the ratio used to set the definition of more compact texture pieces ("Cost Smoothness Ratio"), the maximum size of a single texture ("Max Texture Size"), the version from which the photos are to be used to texture the model ("Texture From") and the colour used for triangles that are not covered by any photo ("Empty Colour").

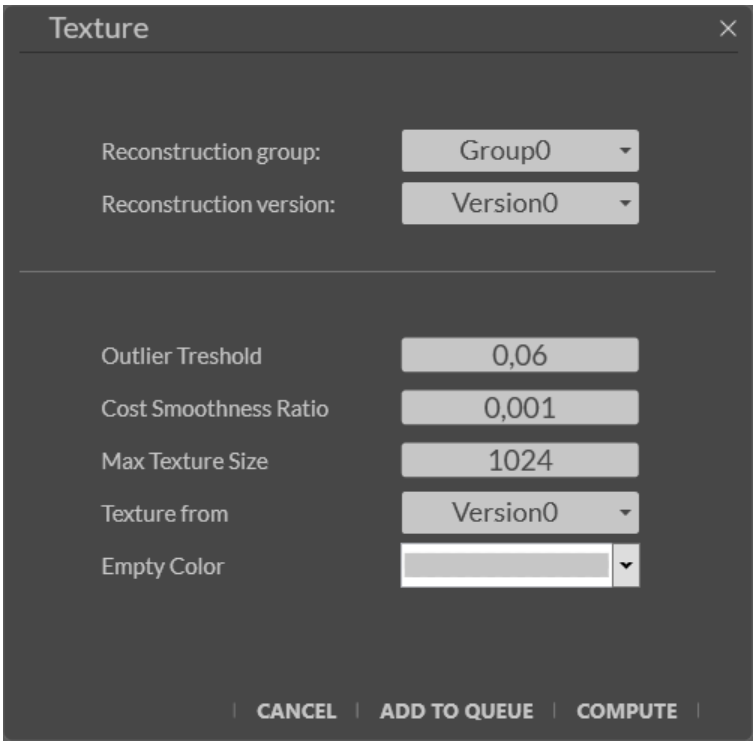

Fig. 27 Texture creation panel

Once all options have been customized, click the "COMPUTE" button to start the model texturing process. The progress of the process will be visualized in the panel with the progress bar (Fig. 28). The duration of this step depends on the mesh density of the model, the resolution and number of photos, as well as the selected options.

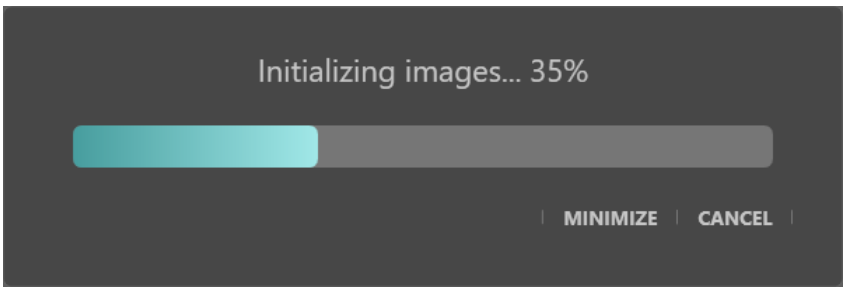

Fig. 28 Texture creation

After successful reconstruction of this stage, the model with the texture should automatically load and display (Fig. 29). Changes such as moving and rotating the object can be made at this stage.

Fig. 29 Textured model

8. Reconstruction queue

Each of the stages can also be added to the reconstruction queue, which can be run later to start the reconstruction stages according to the defined sequence. This enables to define the parameters allowing to reconstruct more than one group. To do so, click the "ADD TO QUEUE" button in the panel of a given stage (Fig. 30).

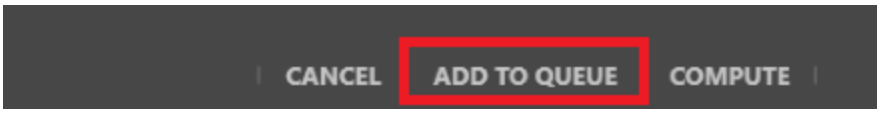

Fig. 30 Add to queue button

In order to run the queue created in this way, in the main menu click the "Reconstruction Queue" button under "RECONSTRUCTION" tab (Fig. 31).

Fig. 31 Reconstruction queue button

After selecting this option, the reconstruction queue panel will be displayed (Fig. 32). The sequence of all stages can be changed by means of arrows - you can remove a given stage from the queue with the red cross or edit individual stages after clicking on the gear icon.

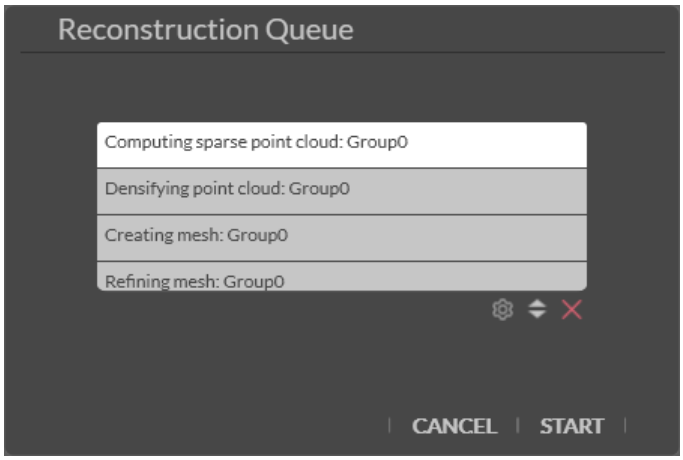

Fig. 32 Reconstruction queue panel

Once all of them have been adjusted, click the "START" button to start the reconstruction queue. The progress of each process will be visualized in the panel with the progress bar. The duration of the whole process depends on the number of queue elements, as well as their settings.

9. Full reconstruction

You can also run the entire reconstruction of a single group and version, from sparse cloud through texturing the model - all in one panel. To do this, in the main menu click the "Full Reconstruction" button under the "RECONSTRUCTION" tab (Fig. 33).

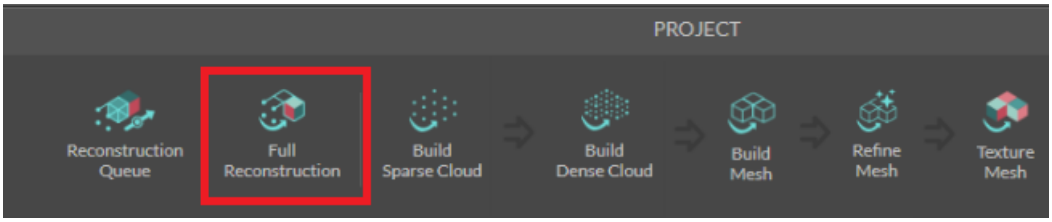

Fig. 33 Full reconstruction button

After selecting this option, the full reconstruction panel will be displayed (Fig. 34). At the top of the panel, select the group and version to be reconstructed. Then you need to customize the options for each stage of the reconstruction or select a configuration preset that will automatically change everything. You can also uncheck each of the stages, which will skip them during the reconstruction process.

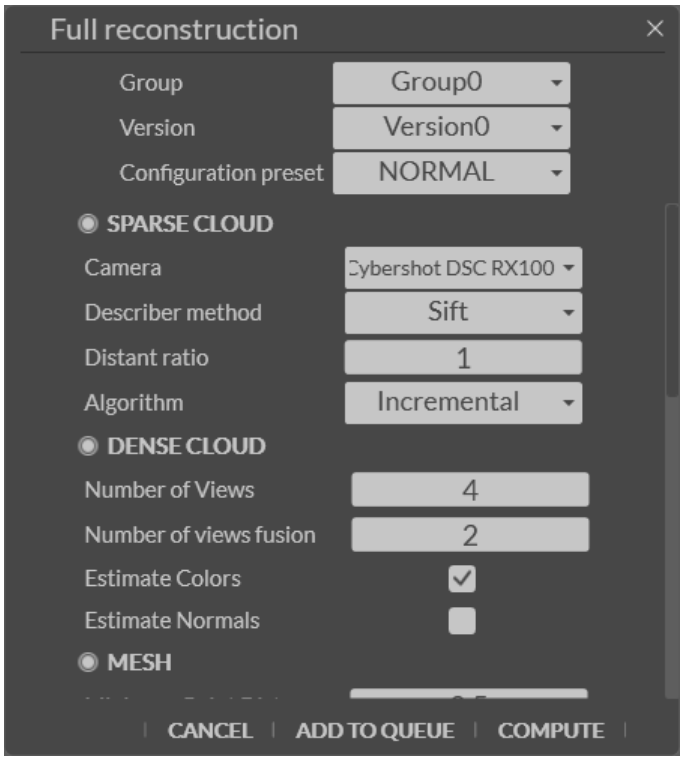

Fig. 34 Full reconstruction panel

Once everything has been customized, click the "COMPUTE" button to start the full reconstruction or "ADD TO QUEUE" to add this process to the queue. The progress of each process will be visualized in the panel with the progress bar. The duration of the whole process depends on the number of photos and selected stages, as well as their settings.

10. Matching point clouds

At the stages of dense and sparse clouds it is also possible to match fragments of reconstructed clouds. To do this, load both clouds you want to match, then select the common area and in the main menu, in the "RECONSTRUCTION" tab click the "Match" button (Fig. 35).

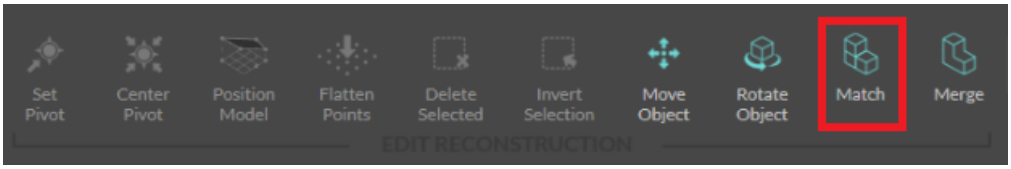

Fig. 35 Point cloud match button

This will display a matching panel where you can select a matching method. After selecting it, click "CONFIRM" which will automatically adjust the clouds relative to each other (Fig. 36).

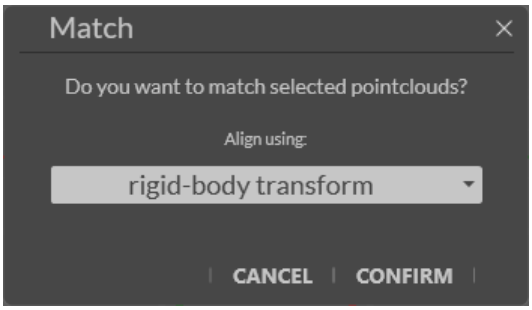

Fig. 36 Point cloud match panel

11. Point clouds merging

Clouds that are matched manually or using the matching panel can be merged into one. To do this, in the main menu, in the "RECONSTRUCTION" tab, click the "Merge" button (Fig. 37).

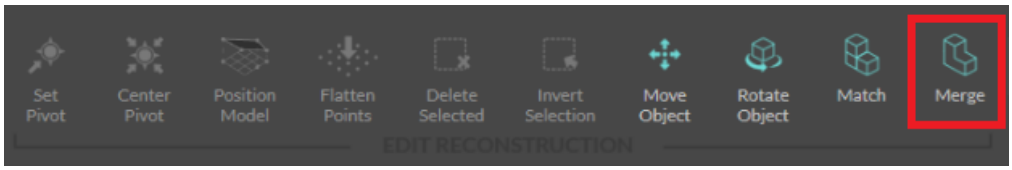

Fig. 37 Point cloud merge button

This will display the merging panel where you can select the type of clouds you want to merge, as well as the groups and source versions (Fig. 38). After selecting the appropriate options, click "MERGE" to merge the clouds or "ADD TO QUEUE", which will add merging to the reconstruction queue.

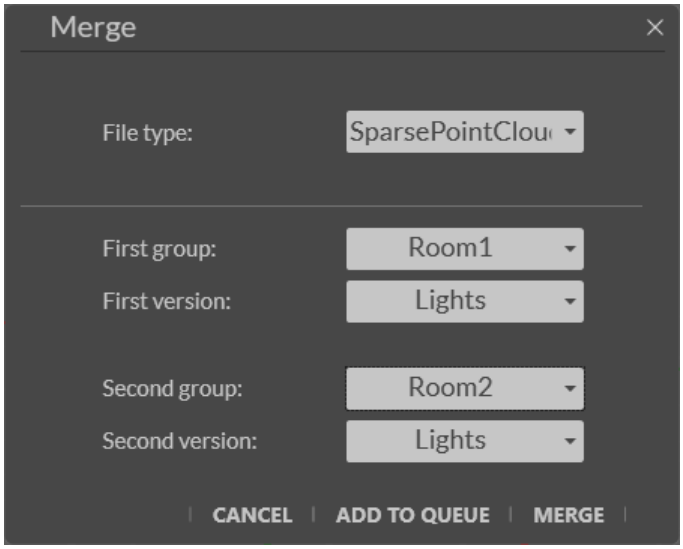

Fig. 38 Point cloud merge panel

13. Exporting the model

After loading and displaying the model it is possible to export it to '.obj' or '.ply' file format. In order to do so, in the "PROJECT" tab click the "Export" button (Fig. 39).

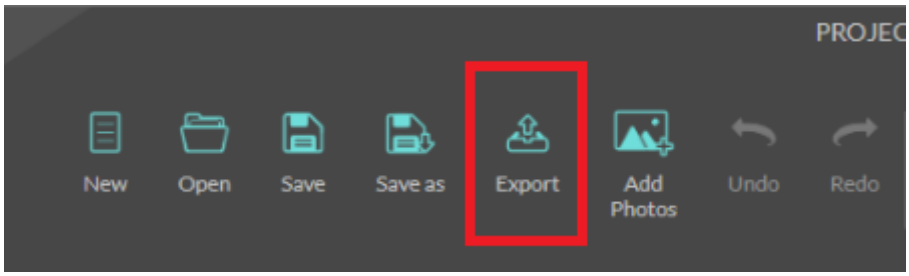

Fig. 39 Reconstructed model export button

This will display the Windows "Save As" dialogue. Here you should select the target directory and the name of the exported model, as well as the file format by choosing the appropriate extension '.obj' or '.ply'. Then click the save button to start the export process, which will be visualized in the panel with the progress bar (Fig. 40). The duration of this process depends on the number of points in the case of the cloud, or the number of vertices in the case of the model, and the size of the texture, if generated.

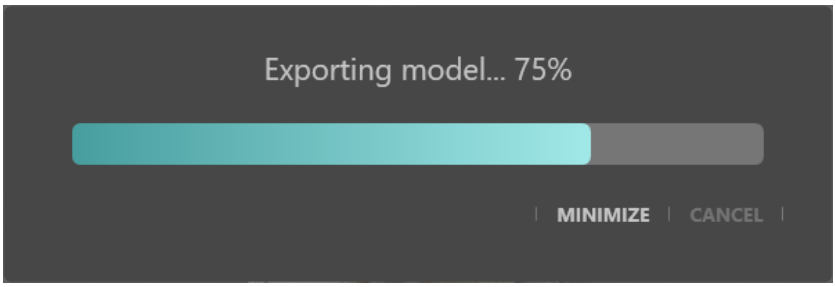

Fig. 40 Model exporting

14. Exporting/importing groups

It is also possible to export groups when a part of the room has been reconstructed in one project and there is a need to move it to another project. To export groups, right-click on the target group and select "Export" (Fig. 41).

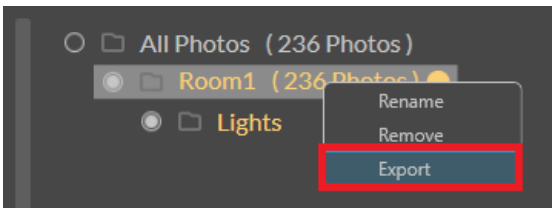

Fig. 41 Group export button

This will display the Windows "Save As" dialogue. Here you can select the target directory and the name of the exported group. Then click the save button to start the export process, which will be visualized in the panel with the progress bar (Fig. 42). The duration of this process depends on the number of photos, the degree of reconstruction of the group and the size of the files generated during the reconstruction.

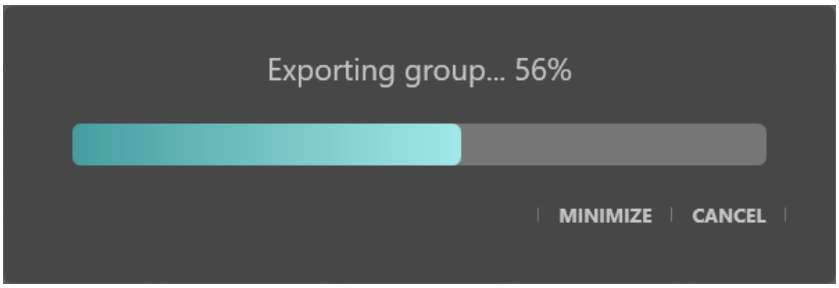

Fig. 42 Group exporting

Once you have successfully exported the group, you can import it to another project. To do this, open the target project, then right-click on "All Photos" and select the "Import" option (Fig. 43).

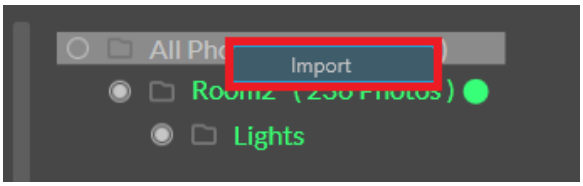

Fig. 43 Group import button

This will display the Windows "Open" dialogue. At this point go to the folder where the group was previously saved, select the file of the exported group and click open. This will start the import process, which will be visualized in the panel with the progress bar (Fig. 44). The duration of this process depends on the size of the group file.

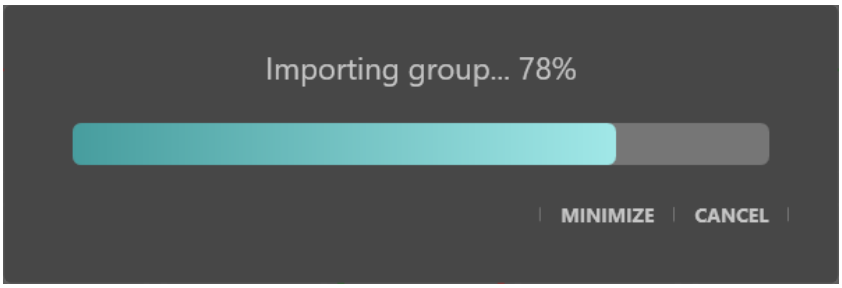

Fig. 44 Group importing

15. Model viewer

The central element of the Sterio Reconstruction application is the viewer of the reconstructed models (Fig. 45). Here you can see the effects of each stage of the reconstruction, from a sparse cloud to a model with texture.

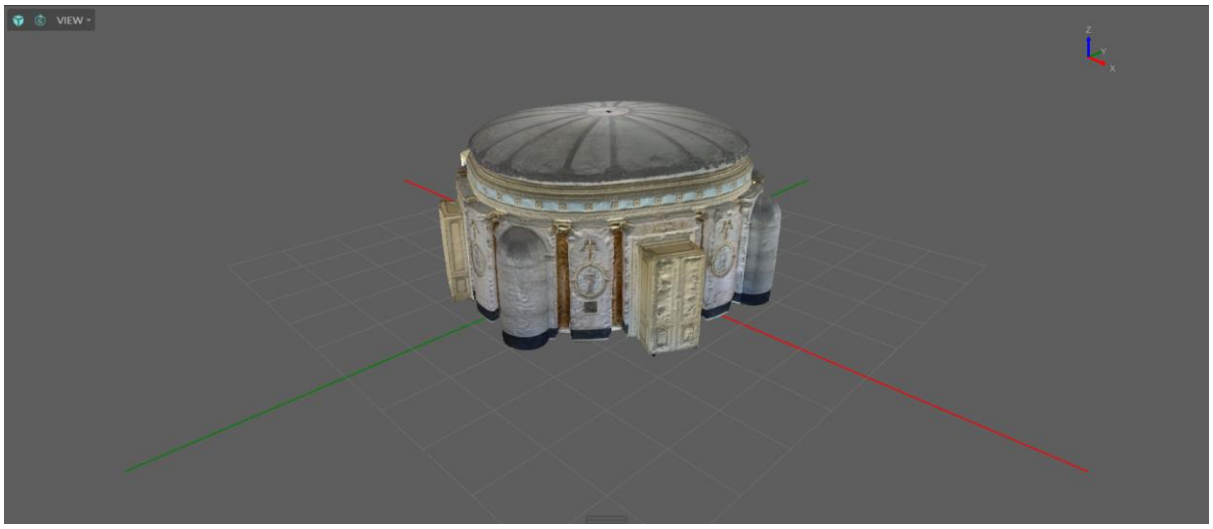

Fig. 45 Model viewer

The camera is controlled by the mouse, hold down the right mouse button to rotate the camera around the model or use the mouse wheel to zoom in and out of the camera. You can also change the camera view using the options in the "VIEW" drop-down menu on the left (Fig. 46). You may also change the camera type to a camera in perspective or a camera in axonometry by using the button on the left-hand side of the "VIEW" menu.

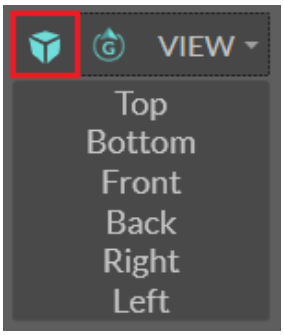

Fig. 46 Camera view options

On the right side of the model viewer there is an indicator of the directional axes marked with their respective colours (Fig. 47). The X-axis and Y-axis are also shown on the grid that is displayed at point (0, 0).

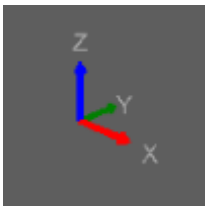

Fig. 47 Directional axis indicator

The mesh serves as a reference point, with the help of which the position, rotation and scale of the model can be verified. In order to switch the mesh on/off, click the "Show Grid" button in the "PROJECT" tab (Fig. 48).

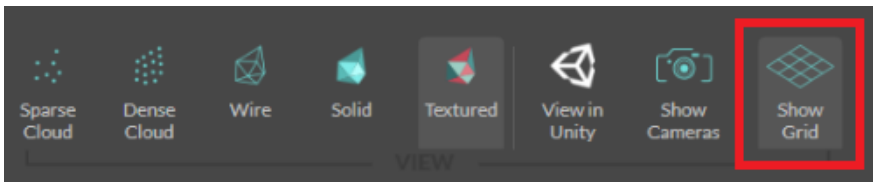

Fig. 48 Grid display button

16. Model editing

You may also perform simple editing of clouds and models in the model viewer. Just select the model, cloud or part of the cloud points, and then select the appropriate option in the "RECONSTRUCTION" tab (Fig. 49).

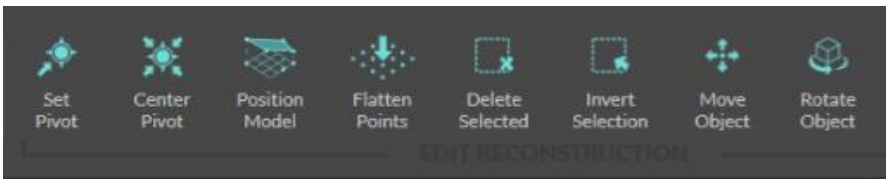

Fig. 49 Cloud/model editing buttons

The available options are:

- Manually setting the model/cloud pivot ("Set Pivot")
- Automatic model/cloud pivot centering ("Center Pivot")
- Positioning model/cloud relative to mesh ("Position Model"), which is done by indicating the plane using 3 points on the model/cloud
- Flattening of cloud points ("Flatten Points"), which is done by selecting a group of points with the mouse and then indicating the plane with 3 points to which it is to be adjusted
- Deleting selected points ("Delete Selected") is used to manually correct errors and noises caused by the reconstruction
- Inverting the cloud points selection ("Invert Selection")
- Moving the model/cloud ("Move Object")
- Rotating the model/cloud ("Rotate Object")