



## **Dynamix mapper**

System for creation of high-precision aerial 3D maps using SLAM

# **User Manual**

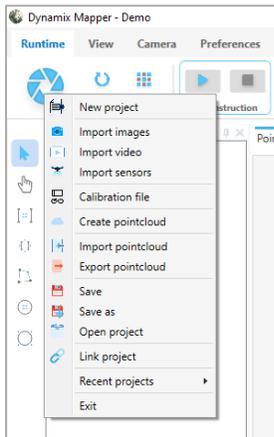
## Contents

Mapping .....	2
Import video.....	2
<i>Sample video</i> .....	3
<i>Import images</i> .....	4
<i>Start mapping</i> .....	4
<i>Manipulate keyframes</i> .....	4
<i>Create point cloud from keyframe graph</i> .....	5
<i>Save project</i> .....	8
<i>Open project</i> .....	8
Georeference.....	8
Analysis .....	10
<i>Move/rotate point cloud</i> .....	10
<i>Manipulation: Delete points, Undo deleted points, Split PC, Combine</i> .....	10
<i>Selection of points</i> .....	11
<i>Drawing lines, quick profiles and measurements tool</i> .....	12
<i>Area and volume calculation</i> .....	14
<i>Elevation profile</i> .....	14
Contours.....	15
Mesh.....	15
Google maps overlay .....	15
Advanced users options.....	16
<i>Camera tab</i> .....	16
<i>Preferences tab</i> .....	16
<i>Keyframe manipulation tools</i> .....	17
<i>Move/rotate point cloud</i> .....	17
<i>Filters</i> .....	18
<i>Densifier section</i> .....	19
<i>Merge two Dynamix Mapper projects</i> .....	19

## Mapping

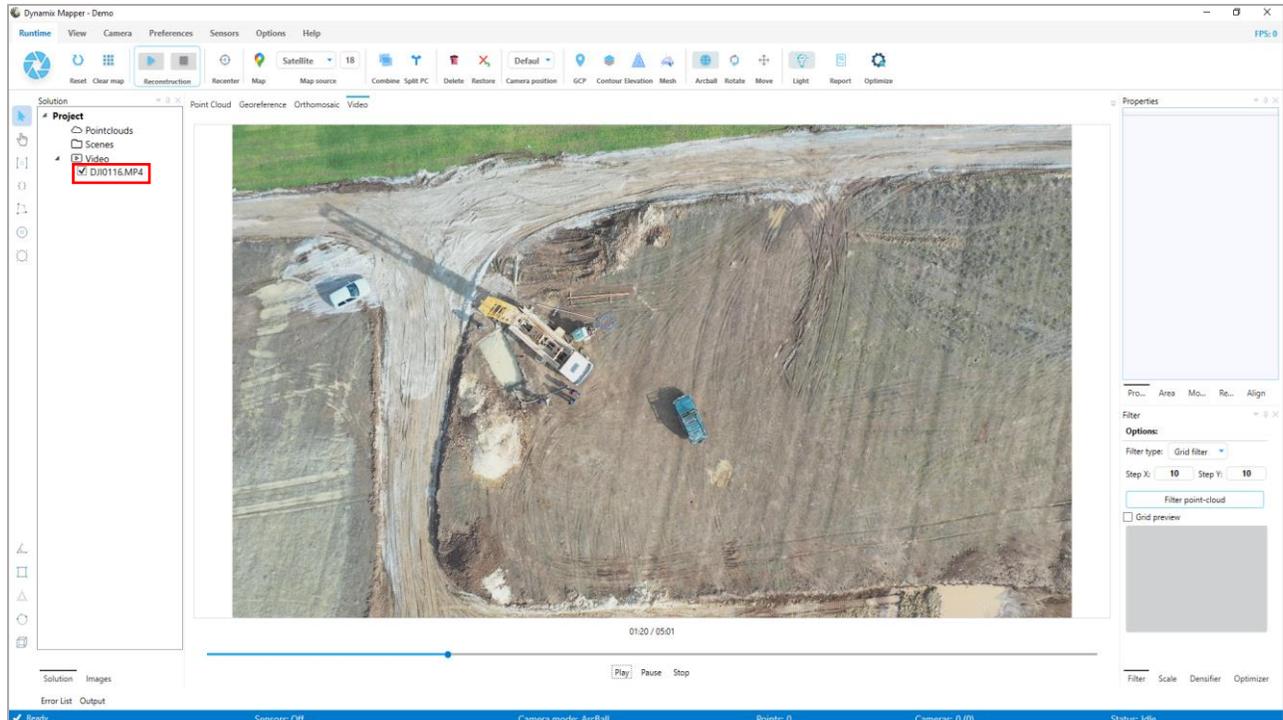
### Import video

Click the Main icon and select “Import Video”.

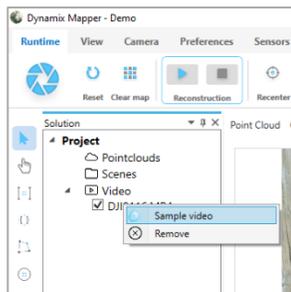


Select the video and Click Open. The video will appear in the Video Folder. Double click on the name will open the video in the video tab where you can play it and zoom in/out.

More videos can be imported in the same project. List with imported videos will be shown under the "Videos" folder.

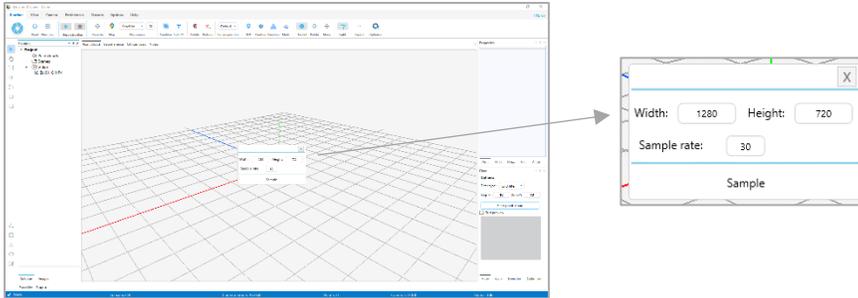


### Sample video



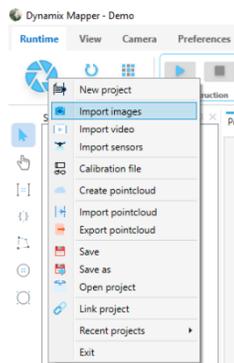
Select the video / videos you want to sample, right click on the name and choose "Sample video" option.

After that, new window will be shown. In that window, set the desired resolution, "Sample rate" value and click the "Sample" button. When the procedure is finished, find the images in the video's folder.



### Import images

Click the main icon and choose “Import images”.

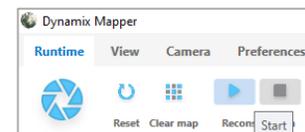


Navigate to the folder of the previously sampled video, find the images folder<sup>1</sup> and click “Select folder”.

### Start mapping

When everything is ready, click the “Start” icon to start mapping procedure. You can stop the procedure anytime using the “Stop” button.

If you are not satisfied with the result, make sure that you are using the right resolution and calibration parameters (check the Camera tab and Preferences tab parts of this manual)



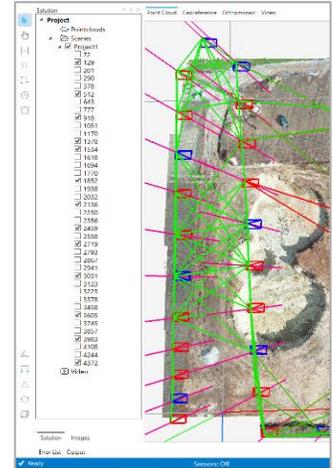
### Manipulate keyframes

After you start the mapping procedure, software will start creating keyframes and the keyframe list will be shown in the Scenes Folder.

There are many keyframe manipulating options:

<sup>1</sup> Images folder will be named regarding this format : Videoname.extension\_WidthxHeight\_Images, example: DJI\_0087.MP4\_1920x1080\_Images

- Show/hide key frames
  - Check/uncheck the box in the keyframe list to show / hide a keyframes. Active keyframes have blue color in the graph, the rest are red.



### Create point cloud from keyframe graph

The recommended procedure for creating a point cloud includes four steps:

- *Manual cleaning*
- *First Optimization*
- *Keyframe overlaps removal*
- *Final (second) Optimization*
- *Point cloud*

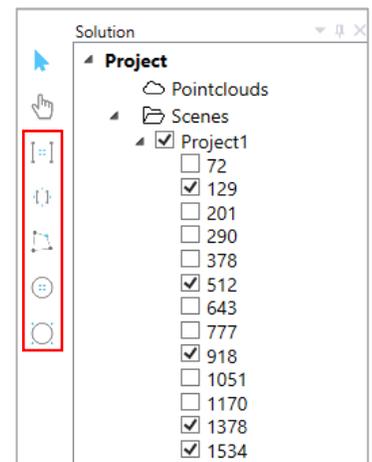
- Manual cleaning

Select all keyframes you want to include in the point cloud. Define the area of interest with manually cleaning the parts that are obviously "noise" (rough cleaning, no precision is needed). In the same time, this will speed up the optimization process (next step).

Clean the graph using the selection tools, included in the general toolbar positioned on the right side of the workspace. There are rectangular, polygonal and circle formats for inner/outer selection. Activate one of the tools and click and drag to create a selection, press right click to confirm.

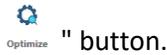
Selected points will become red, click the "Delete" icon to erase the selection.

Cleaning procedure can be repeated as much as it's needed.



- **First Optimization**

When you are done with manual noise cleaning, start the first optimization using the "Optimize



This procedure will clean the rest of the noise, after scanning selected keyframes. This will be base for final optimization before creating a point cloud.

Inspect the result when the procedure is finished. At this stage you can a delete parts, remove or add keyframes. If you want to add / replace keyframes, deselect all keyframes and select the ones you want to add. Repeat the procedures for the new keyframes.

After you finish, select all keyframes you want to use.

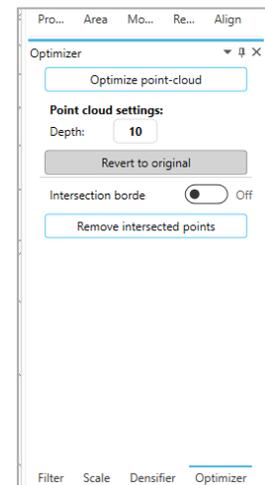
- **Keyframe overlaps removal**



After the First optimization, Keyframe overlaps removal is needed, to reduce the number of points and delete the unnecessary parts.

Go to "Optimizer" tab, located in the right corner, and click "Remove intersected points".

Intersection points (overlapping parts) will become red. Click "Delete, to remove the selected points.



- **Final (second) Optimization**

After cleaning the Keyframe overlaps (intersected points), graph is ready for final optimization.

In the "Optimizer" tab, set the "Depth" parameter (this parameter defines the quality) and click "Optimize point cloud".

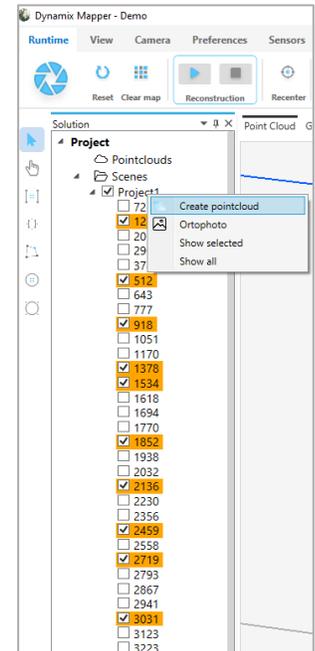
Click "Revert to original" button to undo the changes caused by Optimization.

Inspect the result after Optimization finish. If you are satisfied with results, continue with the next step (create point cloud) or set different value as "Depth" parameter to increase/ decrease quality.

- Create point cloud

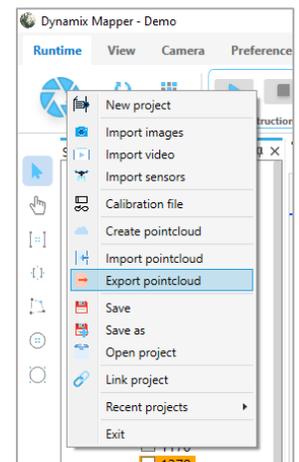
Create a point cloud with all used keyframes or divide the reconstructed area to more parts (point clouds) with different keyframe combinations.

Select the keyframes you want to include to the point cloud. Right-click on the “Project1<sup>2</sup>” and choose the “Create point cloud” option. When the point cloud is created it will be shown in the Point clouds Folder.

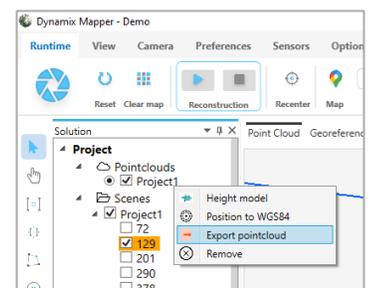


There are two ways to export the point cloud:

- Activate the point cloud with checking the box beside the point cloud’s name, click the Main icon and click “Export point cloud”. Choose the export format, define the export destination, and click “Save”.



- Right click on the point cloud layer and choose “Export point cloud” Choose the export format, define the export destination, and click “Save”.

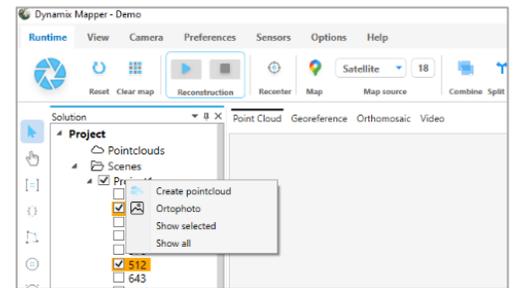


<sup>2</sup> If your project is already saved, “Project1” will be replaced with the name of the saved project.

### ○ *Create Orthophoto*

Create an Orthophoto with all used keyframes or divide the reconstructed area to more parts with different keyframe combinations.

Select the keyframes you want to include to the raster. Right-click on the project's name ("Project1") and choose the "Orthophoto" option. Go to "Orthomosaic" tab to open the created Orthophoto.



There are more options in "Orthomosaic" tab:

- "Save 
- "Rotate 
- "Calculate referent point 
- "Line drawing tool 
- "Clear Orthophoto 



### *Save project*

Click the Main icon and select "Save project", define the location, write the project name and click "Save".

### *Open project*

Click the Main icon and select "Open project", navigate to the project location, select the .tm3 file and click "Open".

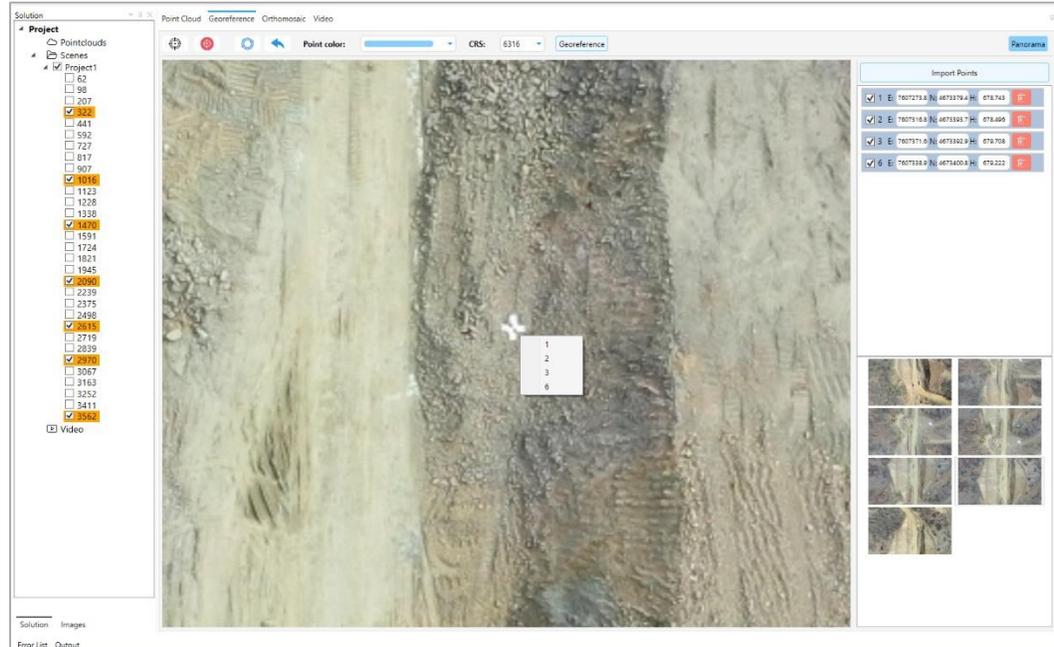
## *Georeference*

### *Ground Control Points*

Use the "GCP" Option to geo-reference the Graph. Use this function when the graph quality is according to your needs.

Click the "GCP 

Use “Import Points” button to import .csv file, or use “Set GCP”  button to create a point with click.



Choose an image, activate “Set GCP”  and use right click to place a point from list, the same could be done with Shift+right click.

If you want to add points manually, use “Set GCP”  and left click, this will place a point and add a row to the list on the right side of the window. Enter the point coordinates there. Do this for all points you want to use for georeferencing.

There are “Reset” , “Undo”  option and “Delete point”  option. When you define the point position once, it will be added to all images that contains that point and every point position can be corrected using “Move point”  button or CTRL+left click (drag to move the marker to desired position).

When you are done with points placement click “Georeference” to finish the procedure. When you click “Georeference” the graph will be changed according to the used points, calculation errors will be presented under each point’s coordinate.

Import Points	
<input checked="" type="checkbox"/>	1 E: 7607273.8 N: 4673379.4 H: 678.743 Diff: X: -0.05612 Y: 0.01577 Z: -0.0588 Rmse: 0.0828
<input checked="" type="checkbox"/>	2 E: 7607316.8 N: 4673393.7 H: 678.496 Diff: X: 0.03421 Y: -0.04339 Z: 0.10728 Rmse: 0.12067
<input checked="" type="checkbox"/>	3 E: 7607371.6 N: 4673392.9 H: 679.708 Diff: X: -0.05206 Y: -0.00084 Z: -0.08841 Rmse: 0.1026
<input checked="" type="checkbox"/>	6 E: 7607338.9 N: 4673400.8 H: 679.222 Diff: X: 0.07398 Y: 0.02846 Z: 0.03994 Rmse: 0.08876

## Sensors

Drone’s sensors can be used after finishing the mapping procedure (at any stage of cleaning, optimization etc.) or before starting the procedure:

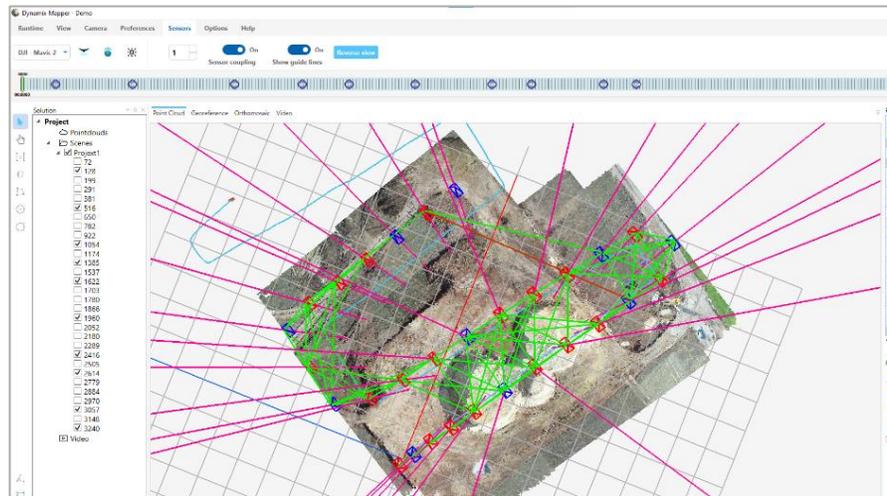
- The first case will result with scaling and positioning the graph to the sensor’s data.
- The second case means that the sensors will be used during the mapping procedure and the result will be according to sensor’s data.

Both cases produce same results.

You need a CSV file that contains sensors data relating to the video time. You can use [this website](#) to convert the original Log file to CSV, after that you need to extract the part of the file related to the video's time interval (find the column "Is video" = 1).

Go to "Sensors" tab and click on the "Load sensors"  icon, select the CSV file and click Open. Time line will be shown after finishing the import step and you can use it to define the desired time/ positions interval that should be processed.

Click the "Position to sensors"  button to place the graph and drone's path to sensor's path. Turn on the "Sensor coupling" to force the generated data to be scaled and positioned according to sensor's data. If you're not sure about the sensor's accuracy, don't use this function.



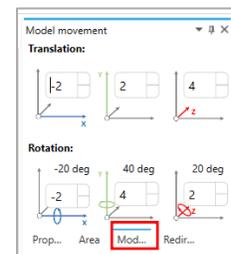
## Analysis

### Move/rotate point cloud



To move / rotate point cloud first, you need to choose which point cloud to edit (if there are more point clouds created or imported), to do that, check/uncheck the circle beside the point cloud's name.

Go to "Model movement" section, where you can find Translation and Rotation tools. Change the X/Y/Z values to move/rotate the point cloud in desired direction.



### Manipulation: Delete points, Undo deleted points, Split PC, Combine.

Manipulation tools are included in the "Runtime" tab.



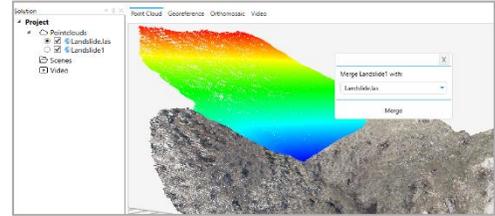
Click on the "Delete"  icon to erase the selected part of the point cloud.

Click on the “Restore ” icon to undo the changes on the point cloud.

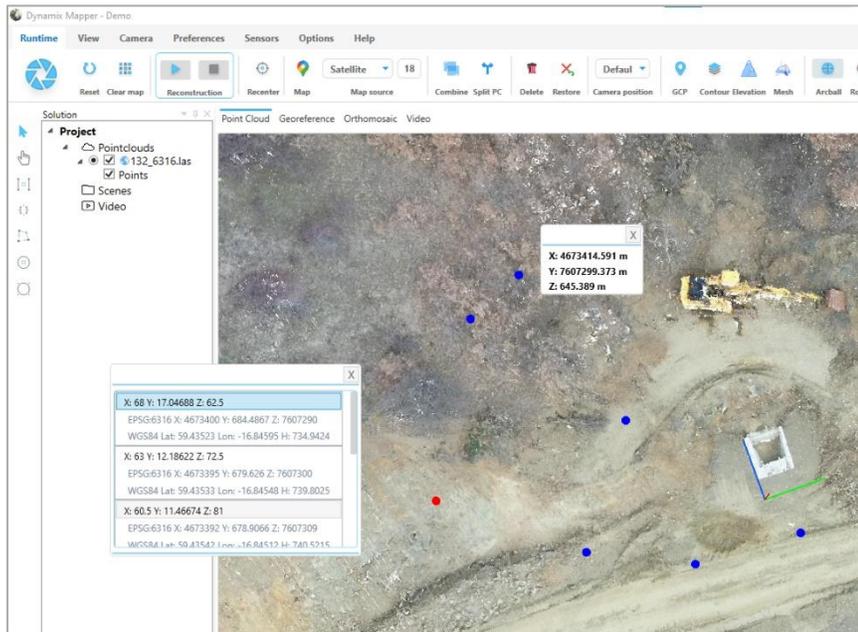
Click on the “Split PC ” icon to split the selected part from the main point cloud.

To merge two point clouds, first you need to activate one of them with checking the circle beside the point cloud’s

name, then click the “Combine ” icon, that will show a window where you need to choose the second point cloud from the drop-down menu, after that click the “Merge” button. This will result with creating new point cloud in the “Point clouds” folder.



points



Selection of

Activate the Selection Point tool and start clicking on the point cloud to select points. When you start selecting points, “Points” layer will be created under

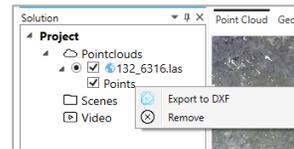
the “Point clouds” folder and pop-up window will be shown with coordinates of the last clicked point.

Check the box beside the “Points” layer to display the list of the selected points with their coordinates.

If you select point from the list, selected entity will become red, click “Delete” if you want to remove it.

Right click on the “Points” layer if you need to:

- Export the elements to .dxf file
- Remove the layer





### *Drawing lines, quick profiles and measurements tool*

Click the “Draw line  ” tool and start clicking or click and drag on the point cloud to draw lines/ polylines, right click to confirm the line/ polyline. This tool will create 3D lines/ polylines, but also quick cross section will be created. When you start drawing, “Lines” layer will be created under the “Point clouds” folder.

Check the box beside the “Lines” layer to display a window that contains the list of the created lines with their lengths and additional tools:

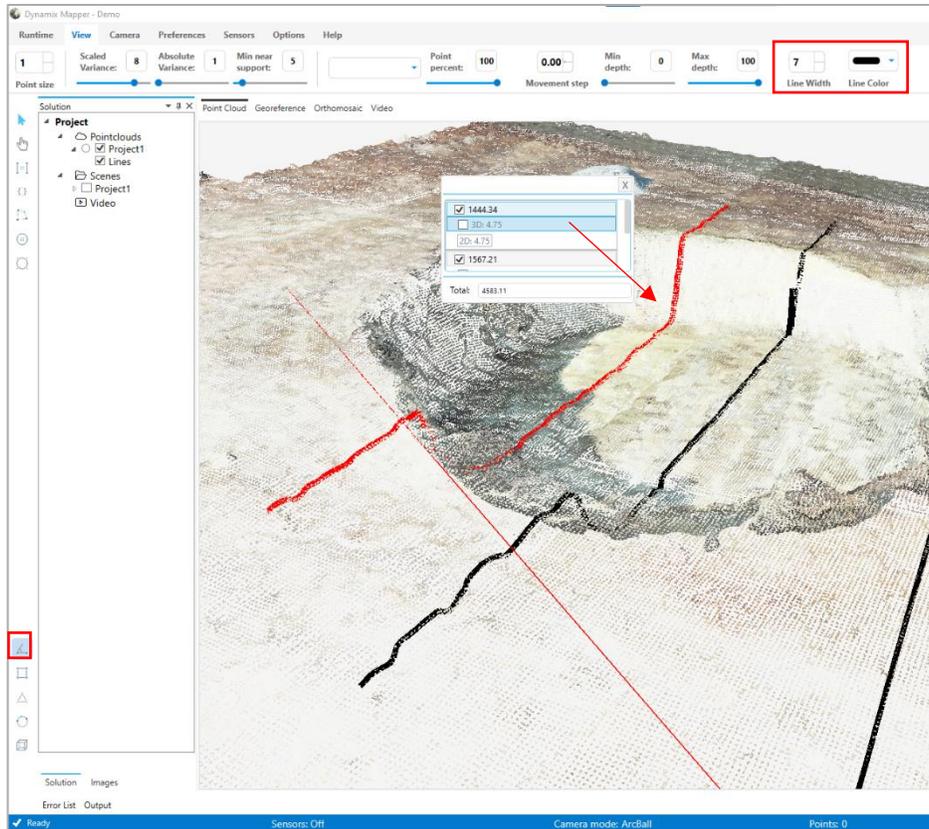
- In the first field there is the length of the cross-section line and turn on/off check box.
- In the second field there is the 3D length of the line (distance from first to last vertex) and turn on/off check box.
- In the third field there is the 2D length of the line.

If you select line/ polyline from the list, selected entity will become red, click “Delete” if you want to remove it.

Right click on the “Lines” layer if you need to:

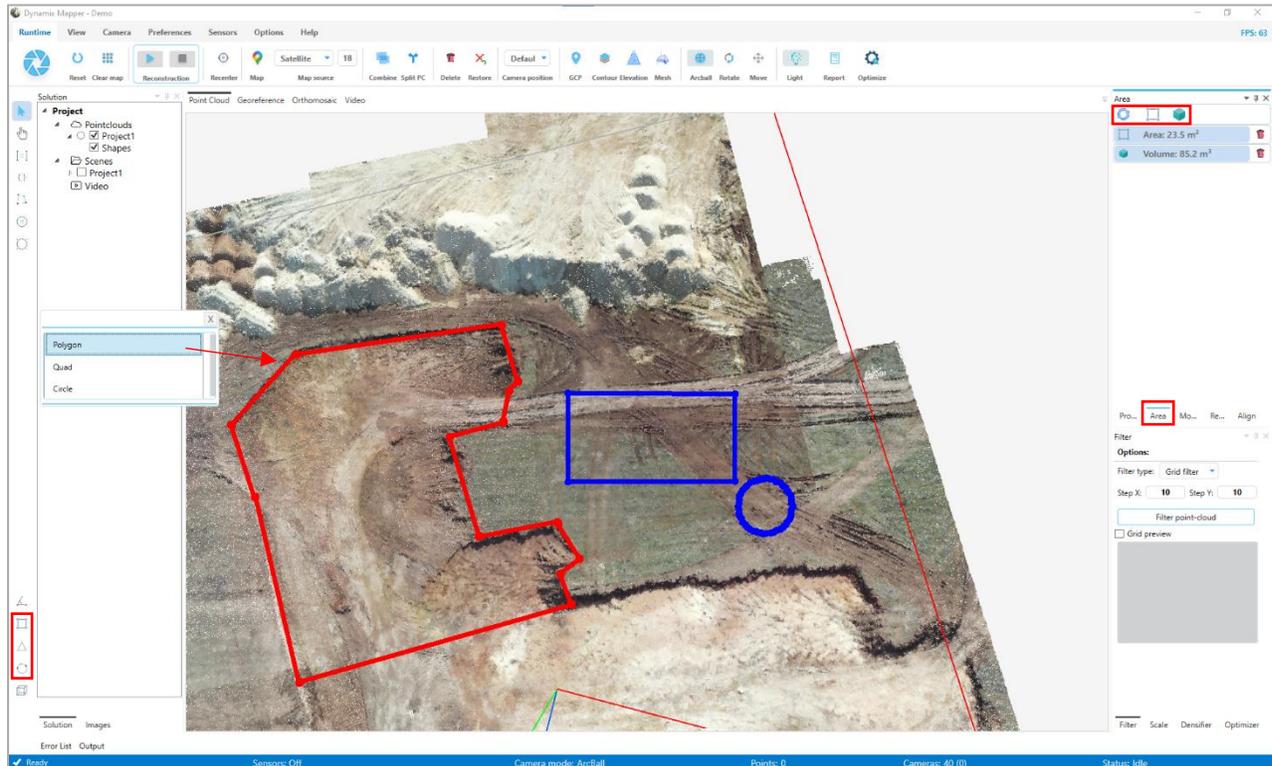
- Export the elements to .dxf file.
- Remove the layer.

Change entities color and width using the options in the “View” toolbar.



### Area and volume calculation

Area and volume tools are included in the general toolbar positioned on the right side of the workspace. There are rectangular, polygonal and circle formats to define the area of interest. Activate one of the tools, start clicking or click and drag to select, right click to confirm, then go to the “Area” section.

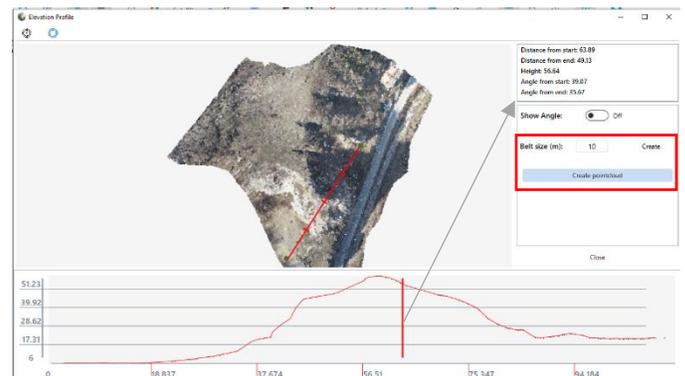


- Use “Reset Selection 

### Elevation profile

- Click the “Elevation Profile  to start drawing the line and click it again to confirm the line.
- Move the “Slide” to read the values on the slider position.

In the “Elevation Profile” window there is an option to create a belt around the line and to cut the Point cloud with it. To use this function, define the “Belt size (m)” and create



it with clicking the “Create” button, after that, click “Create point cloud”. You can find the created point cloud in the “Point clouds” folder.

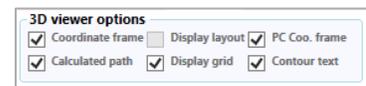
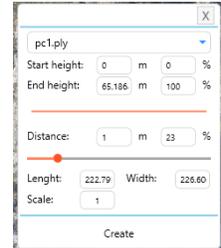
## Contours

Click the “Contour  ” icon, that will open a new window where you need to define creation parameters:

- Choose the point cloud from the drop-down menu.
- Enter the start and end height.
- Define the distance with in the “Distance” field or use the slider.

Click “Create” button to finish the procedure. This will create “Contours” layer under the “Point clouds” folder.

Show/ hide contour elevations using the 3D viewer options from “Options” tab, check/ uncheck “Contour text” checkbox.



Right click on the “Contours” layer for:

- Export the elements to .dxf file.
- Remove the layer.

## Mesh

Click the “Mesh  ” icon, that will open a new window where you need to define creation parameters:

- Choose the point cloud from the drop-down menu
- Define the “Depth”(this parameter act like a resolution and affect the number of triangles), “Width” and “Scale” by entering values in the fields.

Click “Create” button to finish the procedure. This will create “Mesh” layer under the “Point clouds” folder.

Right click on the “Mesh” for:

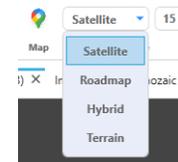
- Export the elements to .obj or .ply format as a batch export or single file.
- Toggle Wireframe
- Remove the layer.



## Google maps overlay

Click on the “Map  ” icon and use the Drop – down menu to choose the base map layer.

To use this function, you need to have imported / created georeferenced point cloud.

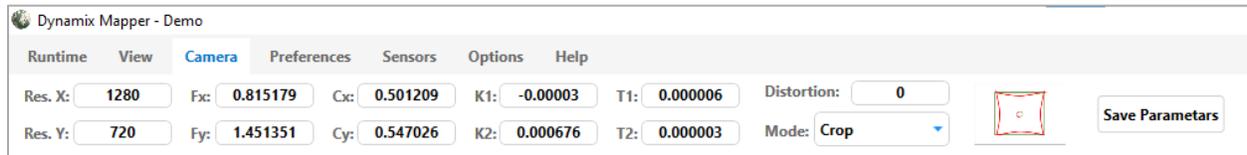


## Advanced users options

### Camera tab

Use this tab if you want to import your own calculated camera calibration parameters. Before edit anything be sure that you have proper values for your camera that means:

- using right format according to the SLAM standards
- calculating the right distortions, only K1, K2 (radial dist.) and T1, T2 (tangential dist.) must be calculated.



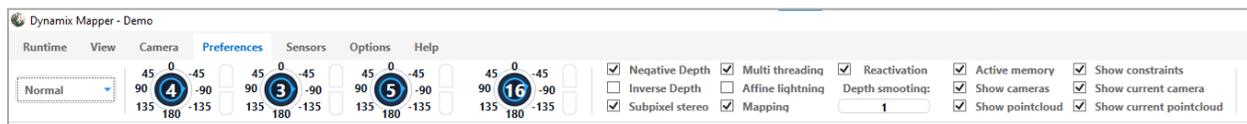
Be sure "Res. X" and "Res. Y" are right values according to the imported Images (*see Sample Video and Import Images parts of this manual (page 3.)*)

### Preferences tab

Preferences tab includes more tools related to the mapping procedure.

There are more options to set up before mapping, like:

- KFUUsageWeight: Determines how often keyframes are taken, depending on the overlap to the current keyframe. Larger value -> more keyframes (first circle, entered value = 4).
- KFDistWeight: Determines how often keyframes are taken, depending on the distance to the current Keyframe. Larger value -> more keyframes (second circle, entered value = 3)
- minUseGrad: Minimal absolute image gradient for a pixel to be used at all. Increase if your camera has large image noise, decrease if you have low image-noise and want to also exploit small gradients (third circle, entered value = 5).
- cameraPixelNoise: Image intensity noise used for e.g. tracking weight calculation. Should be set larger than the actual sensor-noise, to also account for noise originating from discretization / linear interpolation (fourth circle, entered value = 16).



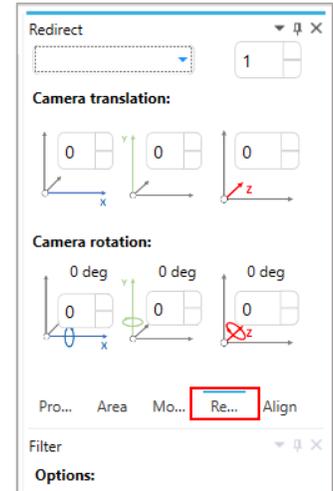
### Keyframe manipulation tools

- Move keyframes

To move a keyframe go to the “Redirect Section, positioned on the right side. Choose a keyframe from the drop-down menu, use the first-row commands. Change the X/Y/Z values to move the selected keyframe in desired axis direction.

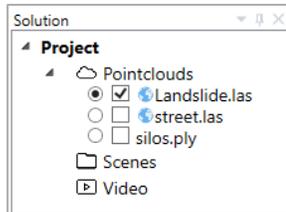
- Rotate keyframes

To rotate a keyframe go to the “Redirect Section, positioned on the right side. Choose a keyframe from the drop-down menu, use the second-row commands. Change the X/Y/Z values to rotate the selected keyframe in desired direction.

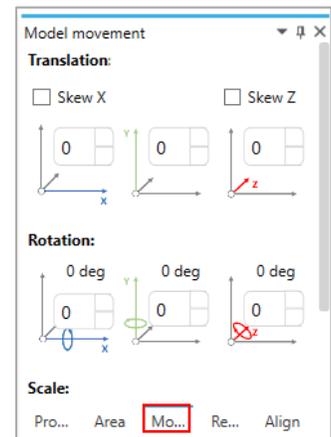


### Move/rotate point cloud

To move / rotate point cloud first, you need to choose which point cloud to edit (if there are more point clouds created or imported), to do that, check/uncheck the circle beside the point cloud’s name.



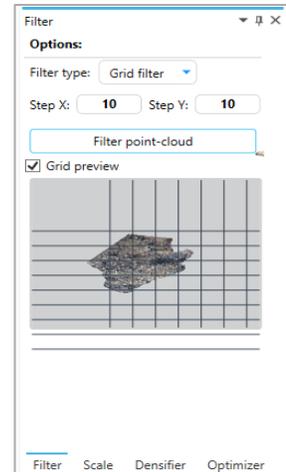
Go to “Model movement” section, where you can find Translation and Rotation tools. Change the X/Y/Z values to move/rotate the point cloud in desired direction.



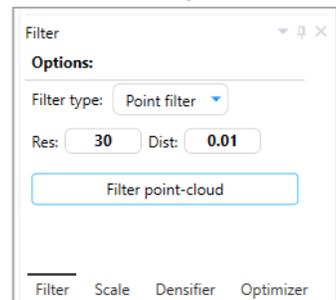
## Filters

Filters are included in the “Filter” section positioned on the right side. You can choose the filter from the “Filter type” drop – down menu.

- The Grid filter is suitable for larger terrain point clouds. In this filter you need to set “Step X” and “Step Y” values, then click the “Filter point cloud” button. The result will be new filtered point cloud in the “Point clouds” folder.

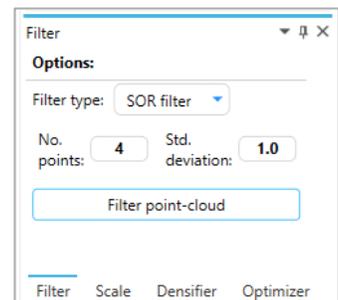


- The point filter is suitable for point clouds with more details. To use this filter you need to enter “Resolution” value, and “Distance” value. Resolution values define the number of parts that would be created during the procedure, the Distance defines the criteria which point to be selected, ex. “Distance: 0.01” will select all points that are not surrounded with other points in radius of 0.01m. Click the “Filter point cloud” button to start filtering the point cloud. The result of this filter will be points selection that meet the criteria. Click “Delete” icon to erase the selected points.



- The SOR filter is suitable for point clouds with more details. To use this filter, you need to enter:

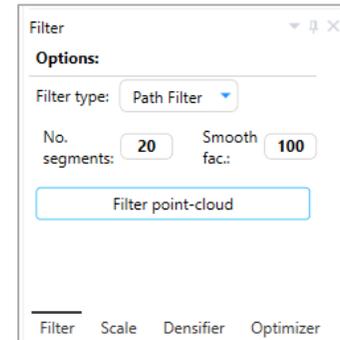
- “No. points” value- the number of neighbors that will be used to compute the 'distance to neighbors' for each point.
- “Std. deviation” value- choose the number of neighbors that will be used to compute the 'distance to neighbors' for each point.



- The Path filter is suitable for filtering and smoothing road surfaces or presenting how the road surface (mostly used for new earthen roads) will look like. This filter changes the point cloud based on the two parameter you need to set and the lines you need to draw as a road border.

Parameters:

- No. segments: this parameter defines how much segments to be created.
- Smooth factor: this is the smoothing coefficient to be applied to each segment, bigger value will produce more smoothest surface.

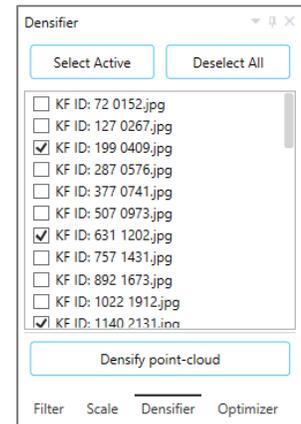


Use the “Draw line ” tool to define the road edges.

### Densifier section

Densifier section is positioned on the right side of the workspace.

Use the “Densify point cloud” tool to fill the gaps after filtering or deleting points. Select the Keyframes that you want to be used as source for filling the gaps and click “Densify point cloud” button. You can select the same keyframes as the ones in the “Scenes” folder by clicking the “Select Active” button, or you can manually select keyframes by checking the box beside the keyframe name.



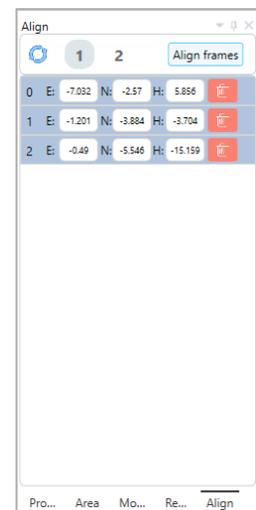
### Merge two Dynamix Mapper projects

To use this option, you need to have at least three common point in both projects. This function can be used in any stage of working, final or first stage, it is not important as soon as you can mark the same three points.

Open the first project (as usual, already described in “Open project” part of this manual), then click the Main icon, choose “Link project”, find the second project and open it.

Go to the “Align” section, click button “1”, active the “Point selection” tool and select the point on the project that you’ve opened first. The point coordinates will appear in the “Align” section. If that project is georeferenced, the coordinates will be according to the project's CRS , otherwise, there would be local (systems) coordinate values. Using the same steps, select the first point on the second project.

Do the same for least three points.



After selecting the pairs of points, click “Align frames” to finish the procedure. This will merge the projects content and everything will act as one project. From this point you can save the content as new project and continue using other functions.