Digital Photogrammetric System

PHOTOMOD UAS

Version 8.1

USER MANUAL

Processing of UAS data (Linux)

RACURS

Table of Contents

1. About

1.1. Purpose of the document

This document contains detailed information about UAS data processing in the *PHOTOMOD UAS* program. The document contains recommendations of processing order, program properties to obtain the best results, and also contains description of additional possibilities when working with UAS data.

1.2. Brief description of system features

The *PHOTOMOD UAS* program is purposed to process data from unmanned aircraft system (UAS). There is a limit on using of the source data when working with program.

Only central projection images with size not more than 100 Mpix could be used a source data.

The following definitions for UAS projects processing are provided in the program:

- preliminary preparing of source images;
- images interior orientation;
- relative orientation of images;
- input and measurement of GCP coordinates;
- Images exterior orientation;
- vectorization;
- stereovectorization;
- building DEM;
- creating orthomosaic;
- creating digital map;
- creating 3d-models.

1.3. Technical Support

The Racurs company technical support provides the actual information about system functionality, characteristics, price and services.

For technical support use one of the following ways:

- e-mail: support@racurs.ru;
- phone: +7 (495) 720-5127;fax: +7 (495) 720-51-28;
- mail: RACURS Co., Ul. Yaroslavskaya, 13-A, office 15, Moscow, Russia.

1.4. Distribution kit

The *PHOTOMOD UAS* program is a stand-alone software, which does not require the *PHOTOMOD* system installation. Also the program could be launched as a module of the *PHOTOMOD* system.

1.5. System requirements

The detailed hardware and software *PHOTOMOD* system requirements are given in detail in “System requirements” in the “General information” User Manual.

Recommended configuration for comfortable work:

System component	Recommended configuration
CPU	*Intel Core i7* or present-day *Intel Xeon* (2.8 GHz or equivalent)
Memory (RAM)	32 GB (minimum 16 GB)
Video adapter	• *NVidia GeForce RTX 3060* (or better), if it is *not* supposed to work in stereo mode • *NVidia Quadro T1000* (or better), if it is supposed to work in stereo mode
SATA (HDD/SSD)	4 TB
Operating system	*Linux* distributions: *Astra Linux 1.8*, *Astra Linux 1.7*, *ALT Linux 10.4*, *RED OS 8.0*, *AlterOS 9.6*
Display	Professional stereomonitors, if it is supposed to work in stereo mode

For the security key, an USB port is required (to use local) or network connection (to use system by network).

RAM

It is recommended at least 16 GB RAM for stable work in the system. If less, processes could take rather more time.

Video adapter

Recommended GPU memory not less than 4 GB.

- *NVidia GeForce RTX 3060* (or better), if it is *not* supposed to work in stereo mode;

- *NVidia Quadro T1000* (or better), if it is supposed to work in stereo mode (*P*, *K*, *M* and *RTX* series can also be used).

Videocards *NVidia GeForce* series do not support page-flipping stereo mode.

Operating system

A hallmark of *Linux* operating system is that performance features of its distributions may differ significantly from each other.

2. Linux pre-configuration and PHOTOMOD UAS installation

Choose one of the following User Manuals:

Table 1. User manuals

User manual	Description
UAS_Astra_Linux_1.7.pdf	*Linux* preconfiguration, *PHOTOMOD UAS* installation (*Astra Linux 1.7*)
UAS_Astra_Linux_1.8.pdf	*Linux* preconfiguration, *PHOTOMOD UAS* installation (*Astra Linux 1.8*)
UAS_ALTLinux_10.4.pdf	*Linux* preconfiguration, *PHOTOMOD UAS* installation (*ALT Linux 10.4*)
UAS_RedOS_8.0.pdf	*Linux* preconfiguration, *PHOTOMOD UAS* installation (*RED OS 8.0*)
UAS_AlterOS_9.6.pdf	*Linux* preconfiguration, *PHOTOMOD UAS* installation (*AlterOS 9.6*)

For more information contact technical support.

3. System first launch

3.1. System configuration

To launch *PHOTOMOD UAS* choose **Start › Science › PHOTOMOD UAS 8.0**.

During the first launch of the system, the message about the required detailed configuration appears. The initial setup of the program can be performed in various ways, depending on the circumstances of the system installation on a particular workstation. The most common situations are the following:

- The system was installed on this workstation for the first time. The user needs to create a folder for storing settings, a resource system, and profiles for organizing local and/or network work. This procedure will be discussed in detail below in this chapter.

- If other *Racurs* software products (*PHOTOMOD*) are already installed and configured on the workstation, the user can quickly connect the installed program to existing profiles and resource systems.

If the connection did not occur automatically during the first launch of the installed program, then in the **Initial setup** window that opens (see below), an already existing **settings folder** used by previously installed software products is indicated;

- The system is installed and configured on several workstations connected to each other via a local network. A new computer is added to the local network where this program was installed for the first time. Connecting such a workstation to an existing network profile is described in “Connecting to existing network profiles” chapter of “General information” User Manual.

3.1.1. Fast system configuration

During the first launch of the system the *PHOTOMOD UAS* initial (fast) setup windows are opened.

If other *Racurs* software products (*PHOTOMOD*) are already installed and configured on the workstation, the user can quickly connect the installed program to existing profiles and resource systems. To do this choose the already existing **settings folder** used by previously installed software products in **PHOTOMOD initial setup** window.

To minimize the process of the program quick setup and go to the **PHOTOMOD initial setup** window (as part of the standard setup of the program), close the quick setup window (or click **Cancel**).

1. Click the ... button to select a physical folder on a local PC to store *resources* of the *PHOTOMOD UAS* projects (see the “Main definitions of resources system” chapter of the “General information” User Manual). Click OK.

   It’s impossible to use logical disk root folder.

   Resources can take up a significant amount of free space on hard drive.

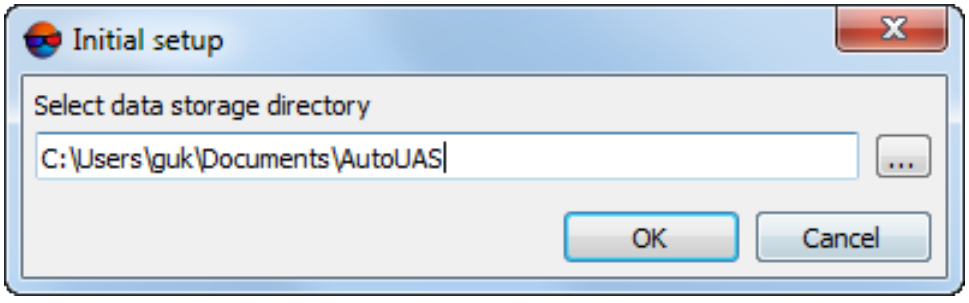


Fig. 1. Choosing a folder to store the projects (an intermediate data)

2. In the **Settings folder** field is displayed path to the *PHOTOMODUAS8.VAR* folder, that is used to store configuration files. Click the ... button to change path to configuration folder or click OK to finish fast system configuration and create the

local profile automatically (see the “Creating local profile” chapter of the “General information” User Manual).

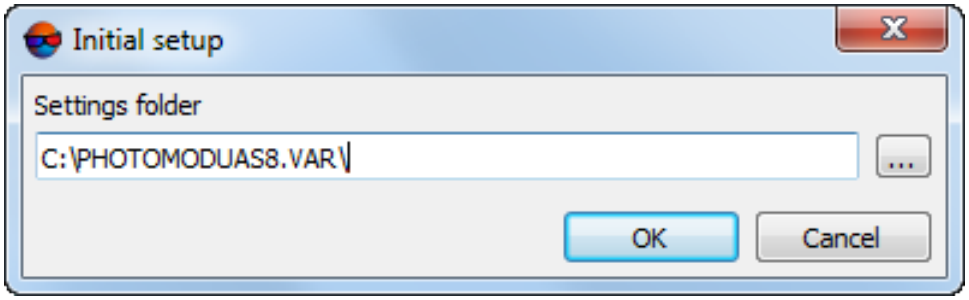


Fig. 2. Choosing a folder to store the configuration files

The *Control Panel* module is used to configure detailed system settings during the further work. More details about program resources, recommendations on organizing of local or network work, and about creating profiles and virtual folders connecting see the “Control Panel. Profiles management” chapter of the “General information” User Manual.

3.1.2. Advanced system configuration

If the existing folder for data storage (or configuration files folder) are not specified during the fast system setup, the message about the required detailed configuration appears. To do this, perform the following:

1. Click OK.

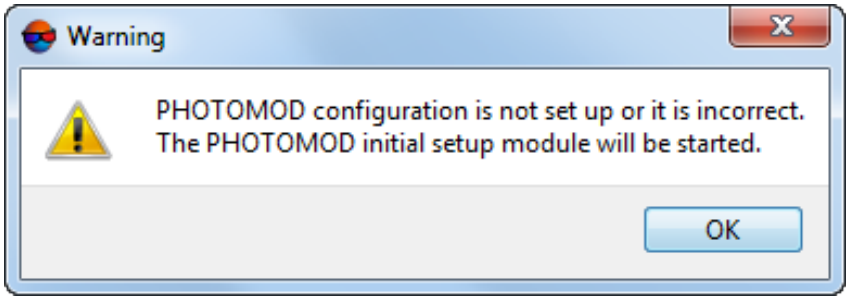


Fig. 3. Information message

2. The **PHOTOMOD initial setup** window opens:

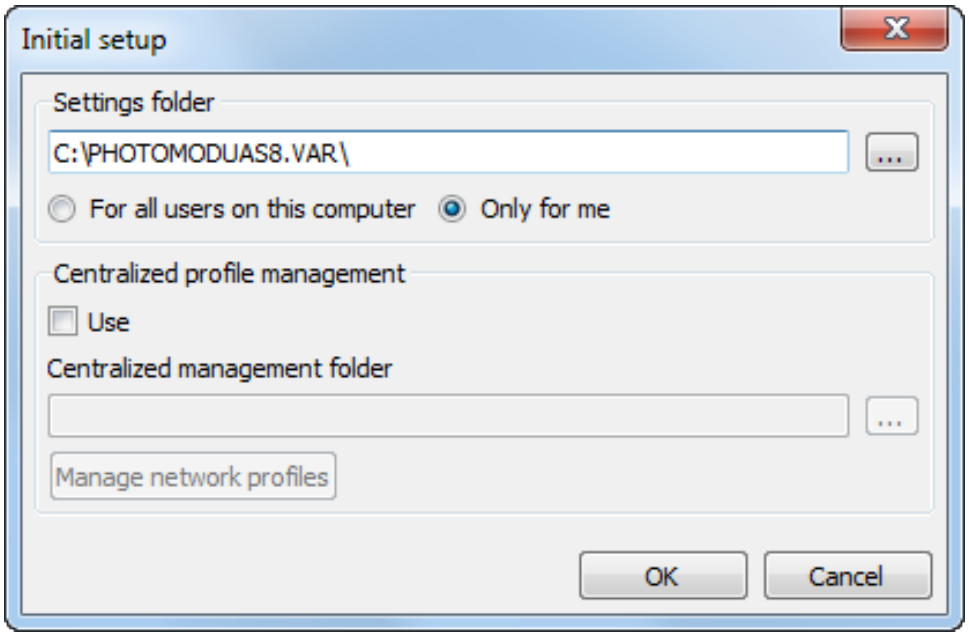


Fig. 4. The PHOTOMOD initial setup window

In the **Settings folder** section is displayed path to the *PHOTOMODUAS8.VAR* folder, that is used to store configuration and temp files. Click the ... button to change path to configuration folder.

If other *Racurs* software products (*PHOTOMOD*) are already installed and configured on the workstation, the system provides for connecting the program to existing profiles and resource systems.

For this, the **settings folder** is to be specified which is used by these software products. In this case, the user will not be required to perform the steps described below to create a new local profile.

Choose **For all users on this computer** to use one configuration folder for all users of current workstation, otherwise choose **Only for me**.

In the **Centralized management folder** section set the **Use** checkbox on and define a folder.

Creation of a *local profile* is described in this Chapter. *Network profiles* management is described in “Creating network profile” chapter of the “General information” User Manual.

3. An info that at least one *local profile* must be created appears:

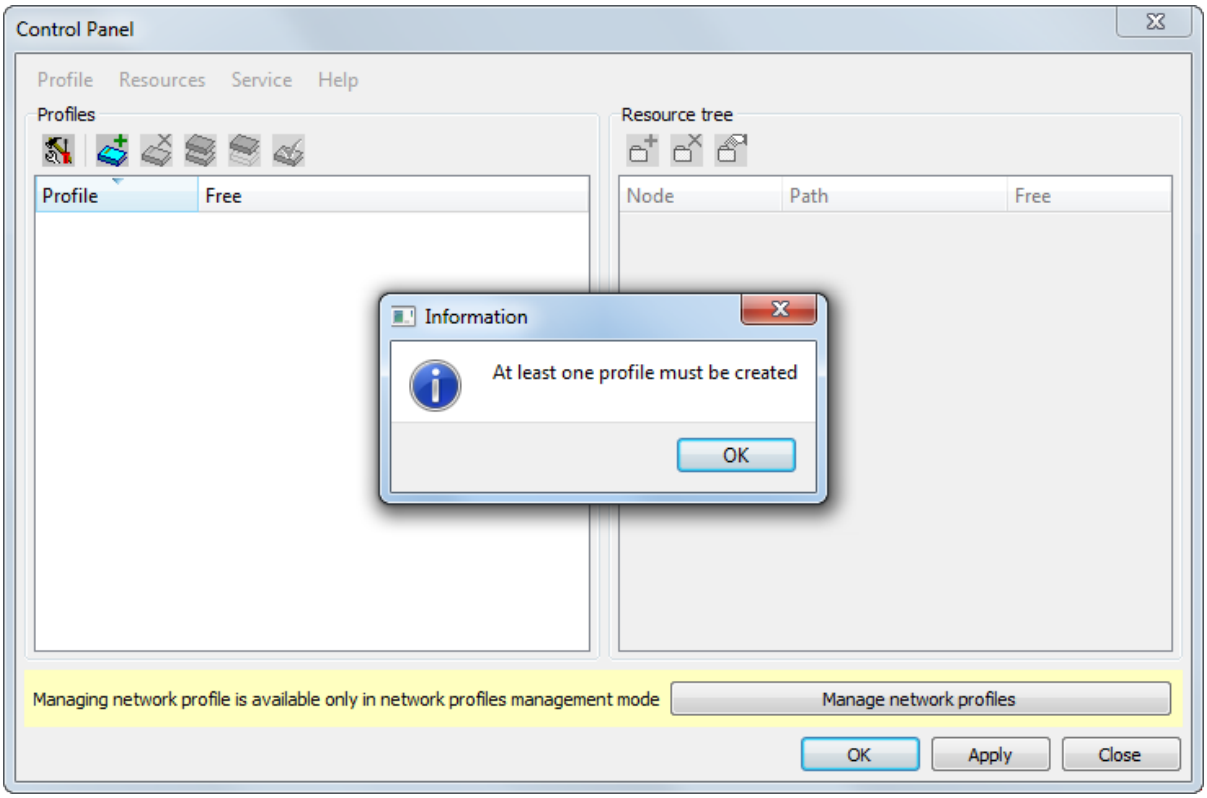


Fig. 5. Information message

Нажмите OK.

4. Specify a *local profile* name:

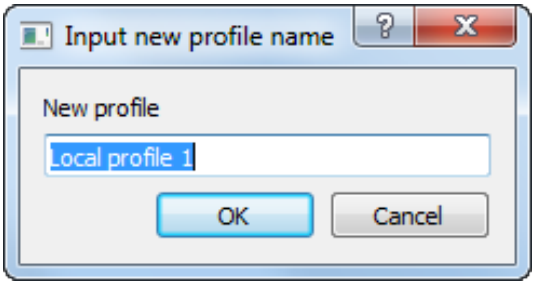


Fig. 6. Local profile name setup

Click OK.

5. Input a **virtual folder name** – arbitrary text is used to identify data in folder.

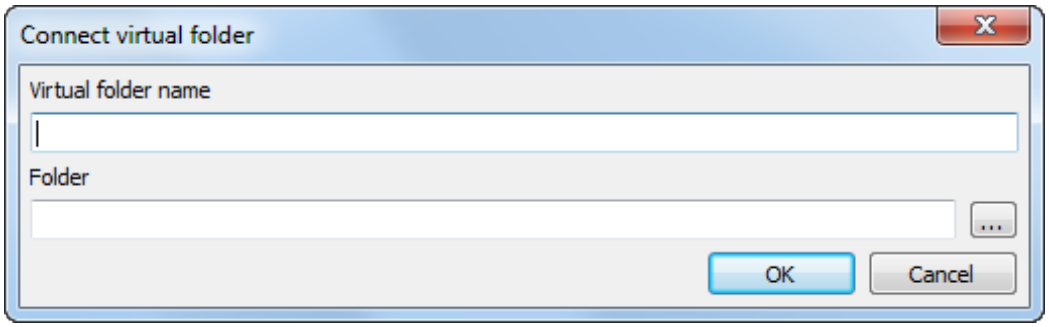


Fig. 7. Connect virtual folder

In the **Folder** section choose a physical space for connecting as a virtual folder:

- choose **Folder** to use only one *network or local* physical folder click the ... button and choose a folder;

  It's impossible to use logical disk root folder.

  To connect folder only read access for this folder is required.

  Local profile folder could be placed both on a workstation, where the system runs, and on any workstation of the network.

  For data management, it's necessary to consider the feature of *Linux* that hard drive partitions, USB drives, network drives, and other data carriers connected to the workstations are to be *mounted*.

  Mounting a file system is a system process to prepare a disk partition for the operating system. As the case may be, this operation can be performed either manually or automatically.

  Data stored on a *connected* but not *mounted* device will not be accessible. Re-mounting of a connected device may be required, for example, after an operating system reset.

  Detailed information about the features of the *Linux* operating systems can be found in the appropriate user manuals, for example, the *Astra Linux* user manual.

- [optional] choose **Storage group** to use several local or network folders as virtual.

  The use of a group of storages is described in detail in "Storages" chapter of the "General information" User Manual.

Click OK.

4. Interface and its elements

4.1. Work area interface

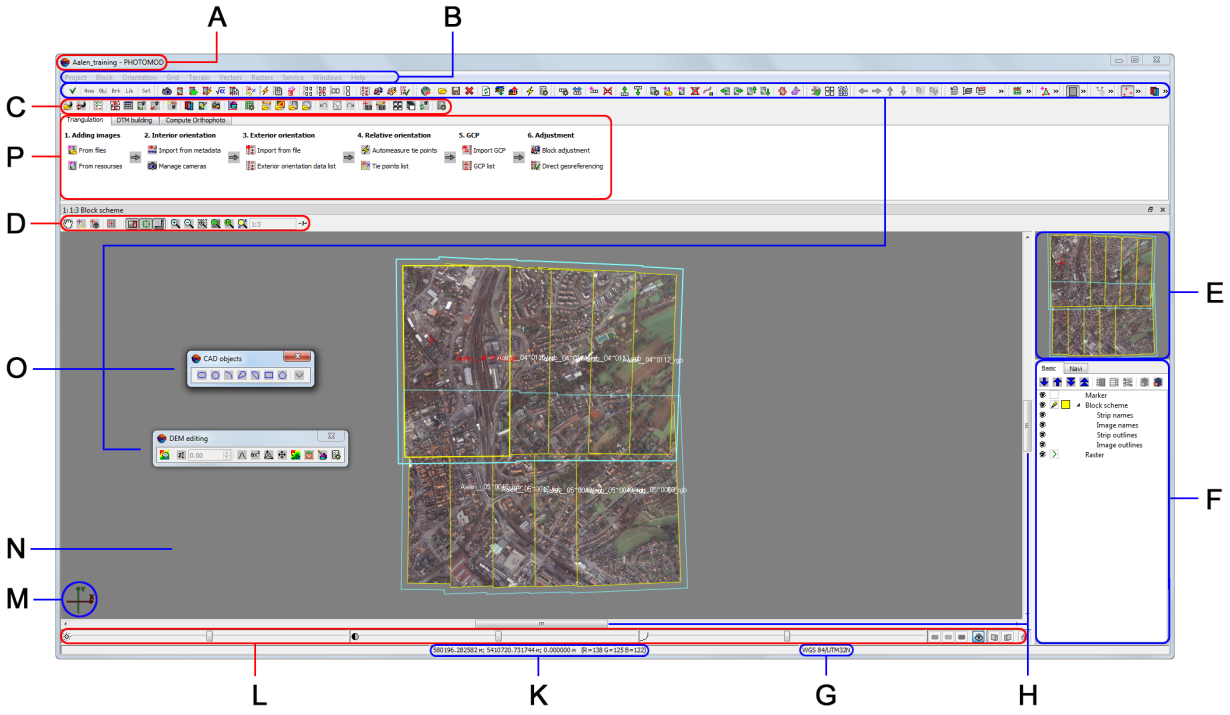


Fig. 8. The main program's window

The GUI contains the following elements:

- title with name of opened project (*A*);
- the main menu bar (*B*);
- the **Aerial Triangulation**, **Compute DEM** and **Compute Orthophoto** tabs (*P*). The tabs layout and content displays the main steps of UAS data processing workflow;
- the main toolbar is used for quick access to main program functions (*C*);
- the optional toolbars is used for quick miscellaneous program functions access (*O*);

The system interface is flexible for customizing the locations of additional toolbars according to the user's needs. Additional toolbars can either be fixed in designated sections of the work area (top or bottom, right or left) or undocked by the user and placed in any place in the 2D window.

Depending on the user-set interface configuration, docked additional toolbars can be partly minimized (some buttons will be hidden). Docked (optionally minimized) subtoolbars are displayed as one line, vertical or horizontal, depending on their location.

By default, additional toolbars are docked at the top of the workspace, to the right of the main toolbar. Additional toolbars are marked with a special icon located on the left or top (depending on the location) edge of the panel. To maximize the toolbar, click (⌄) on the right (at the bottom).

Undocked additional toolbars are always displayed horizontally, in one line, in a fully maximized form. To undock a toolbar (or pinpoint it in any place), move the cursor over the icon and, holding down the **left mouse button**, drag the toolbar to the area of its targeted location (the cursor's shape changes ( ) when it is possible to capture the toolbar).

- the 2D-window, used for data displaying, contains the following elements:
  - the toolbar is used for the 2D-window modes managing (*D*);
  - the work area is used for viewing and processing with loaded data of project (*N*);
  - the navigation bar is used for fast moving on the specified block images area of project (*E*);

    To do this, click on the chosen point in **Navigation bar**. To set the layers visibility, move to the **Navi** tab in the *layer manager*.

  - the *Layer manager* is used for managing of project layers (*F*);
  - the axes direction of project coordinate system (*M*);
  - the status bar is used for viewing current real (ground) and pixel marker coordinates and brightness, contrast and gamma data adjustment in the work area (*G, K, L*);

    GSD value is displayed to the right of the current marker coordinates (excluding radiometric correction windows).

    RGB brightness values are displayed to the right of the current marker coordinates. When placing the marker within the work area, the following brightness values are displayed: `R=128 G=128 B=128`.

  - scroll bar in 2D-window (*H*);
  - the **Status** bar, near the current marker coordinates (*K*), displaying messages of error or success (e.g., after marker was not successfully placed to ground with correlator – `Bad point: correlation error`).

4.2. The main toolbar

The main toolbar is used for quick access to main program functions and also contains **Aerial Triangulation**, **Compute DEM** and **Compute Orthophoto** tabs. The tabs names, layout and content displays the main steps of UAS data processing workflow;

Table 2. Brief description of main toolbar

Buttons	Function
	to open block editor window (see the “Project creation” User Manual)
	to reload project to display the last saved version of project
	to perform import of exterior orientation from metadata
	to open block layout window (see the “Aerial triangulation” User Manual)
	to open block editor window (see the “Project creation” User Manual)
	to display all images, loaded to project, in the **Block scheme** window
	to display only selected on scheme images in the **Block scheme** window
	to load triangulation points
	to open 2D-window for selected stereopair (see the “Vectorization” User Manual)
	to launch the *DustCorrect module* to edit MS-TIFF images (see the “Project creation” User Manual)
	to launch the 3D-mod module
	to create, open and save regular nodes grid (see the “Create DTM” User Manual)
	to load vector file into the project (see the “Vectorization” User Manual)
	to load DEM file
	to load georeferenced raster image to project as a layer
	to load web-map
	to undo the last action (see the “General system’s parameters” User Manual)
	to display the list of last actions (see the “General information about system” User Manual)
	to redo the last undone action (see the “General system’s parameters” User Manual)
	to open the **Marker** window (see the menu description in “Vectorization” User Manual)
	to open the **Measurements** window (see the menu description in “Vectorization” User Manual)
	to arrange 2D-windows evenly
	to stack 2D-windows
	to refresh all opened 2D-windows
	to close all 2D-windows
	to open the general program parameters window (see the “General system’s parameters” User Manual)

4.2.1. The main toolbar tabs

Table 3. Brief description of the Triangulation tab of the main toolbar

Buttons	Function
From files	is used for adding image files, locating in the resources out of active profile, to the selected strip (see the “Adding images from files” chapter of the “Creating project” User Manual)

Buttons	Function
From resources	is used for adding image files, locating in the resources of active profile, to the selected strip (see the “Adding images from resources” chapter of the “Creating project” User Manual)
Import from metadata	to perform import of exterior orientation and interior orientation parameters from metadata (see the “Import orientation from metadata” chapter of the “Aerial triangulation” User Manual)
Cameras	to input/edit parameters of project cameras, and also to assign cameras to project images (see the “Images interior orientation” chapter of the “Aerial triangulation” User Manual)
Import from file	to perform import of exterior orientation data from file of text format (see the “Import of exterior orientation parameters” chapter of the “Aerial triangulation” User Manual)
Exterior orientation data list	to open exterior orientation data list (see the “List of exterior orientation parameters” chapter of the “Aerial triangulation” User Manual)
Automeasure tie points	to setup and start automatic tie points measurement using UAS data (see the “Automatic measurement of tie points coordinates (aerial survey)” chapter of “Aerial triangulation” User Manual)
Tie points list	to open **tie points list** (see the “The “Triangulation points” window” chapter of the “Aerial triangulation” User Manual)
Import catalog	to perform import of ground control points from `*.txt` and `*.csv` files (see the “Import of GCP catalogue” chapter of “Aerial triangulation” User Manual)
GCP list	to open **GCP list** (see the “GCP list” chapter of the “Aerial triangulation” User Manual)
Block adjustment	to open the **adjustment** toolbar (see the “Block adjustment” User Manual)
Direct georeferencing	to perform adjustment using imported exterior orientation parameters (see the “Aerial triangulation” and “Block adjustment” User Manuals)

Table 4. Brief description of the Compute DEM tab of the main toolbar

Buttons	Function
Compute points	to start automatic points calculation (see the “Points automatic calculating” chapter of “DTM Generation” User Manual)
Buildings and vegetation filter	to launch the buildings and vegetation filter *for points* (see the “Buildings and vegetation filter” chapter of “DTM Generation” User Manual)

Buttons	Function
Build TIN	to build TIN by points and breaklines (**Ctrl+N, T**, see the “Building TIN” chapter of “DTM Generation” User Manual)
Build from TIN	to build DEM by loaded TIN (**Ctrl+N, D**, see the “DEM creation by TIN” chapter of “DTM Generation” User Manual)
Build contours	to create contour lines by DEM (see the “” chapter of “DTM Generation” User Manual)
Dense model	to build dense DEM using SGM method (see the “Dense DEM generation using SGM method” chapter of “DTM Generation” User Manual)
Slope based filter	to launch the buildings and vegetation filter *for DEM* (see the “Slope based filter” chapter of “DTM Generation” User Manual)
Fill NULL cells	to restore null cells of DEM using the smooth interpolation method (see the “Filling null cells using smooth interpolation” chapter of “DTM Generation” User Manual)

Table 5. Brief description of the Compute Orthophoto tab of the main toolbar

Buttons	Function
Orthorectification	to display the orthorectification toolbar (**Ctrl+Alt+M**) (see the Orthomosaic building User Manual)
Start GeoMosaic	to start *PHOTOMOD GeoMosaic* program (see the The Orthophotomaps creation User Manual)

4.3. Brief description of main menu

The program main menu contains the menu items for mosaic creation, vector data processing, additional applications starting and setting parameters.

Table 6. Main menu

Menu	Function
Project	contains menu items to create, open, save and convert project, and also to get an information about project
Block	to form images block of mosaic project
Orientation	contains menu items to interior orientation, load and use ground control and triangulation points, exterior orientation parameters and also to import and export triangulation points for various formats
Grid	to create, open and save regular nodes grid (see the “Regular grid of nodes” chapter of the “Create DTM” User Manual)
Terrain	to prepare the base layers for the further DTM/DSM creation: calculating 3D-points, building TIN & tex-

Menu	Function
	tured TIN 3D surface, building DEM, building LAS point clouds, building true ortho, building smooth contours
Vectors	purposed for creating, editing, import/export of vector data (see the "Vectorization" User Manual)
Rasters	to load and georeference raster images, and also to launch additional modules for editing and orthorectification images
Service	to launch applications, load additional data, set general parameters and edit coordinate systems
Window	to open additional toolbars and windows: new 2D-window, Marker and Measurements windows, window of Object Attributes (see "The "Window" menu" chapter of the "General information about system" User Manual)
Help	to start the *"Help"* system

4.4. The "Project" menu

Table 7. Brief description of Project menu

Menu items	Function
New...	to create a new project (see the "Project creation" chapter of the "Creating project" User Manual)
Open/Manage (**Ctrl+Alt+O**);	to open the project management window (see the "Project management window" chapter of the "Creating project" User Manual)
Copy	to create a copy of project selected in the list (see the "Copying project" chapter of the "Creating project" User Manual)
Backup project...	to create project backup (see the "Creating project backup" chapter of the "Creating project" User Manual)
Project export...	exports project data into a `*.xml` file, which is used in *Blocks Exchange XML* protocol between *PHOTOMOD* and external software such as *Agisoft Metashape* or *Bentley ContextCapture* (only for central projection projects, see the "Export and Import of Projects" chapter of the "Creating project" User Manual)
Project import...	imports project data from a `*.xml` file, which is used in *Blocks Exchange XML* protocol between *PHOTOMOD* and external software such as *Agisoft Metashape* or *Bentley ContextCapture* (only for central projection projects, see the "Export and Import of Projects" chapter of the "Creating project" User Manual)

Menu items	Function
Reload	to reloads a project to update it after changes appeared during mutual work
Synchronize	to re-calculate a project data after editing of some parameters, e.g. re-calculates interior orientation after changing camera parameters
Close	to close current project and to start “without project” mode (see the ““Without project” mode” chapter of the “Creating project” User Manual)
Properties	to open the window used to view and edit some parameters of opened project (see the “Project properties” chapter of the “Creating project” User Manual)
Recent	to open a list of projects opened recently in the system
Join projects...	to merge data from several projects of active profile (see the “Joining projects” chapter of the “Creating project” User Manual)
Project state	to display processing status of the following project stages (see the “Viewing project report” chapter of the “Creating project” User Manual)

4.5. The “Block” menu

The **Block** menu, buttons of **Block editor** window’s toolbar and **Block forming** additional toolbar are used to work with images block in 2D-window.

It includes extended features for images block forming, as well as additional functions for block scheme creation and obtaining of auxiliary data.

Some menu items allow to manage operations with selected images in 2D-window. The **Tools** additional toolbar is used for group selection (highlighting) of images block in 2D-window (see the “Vectorization” User Manual).

Detailed information about **Block** menu items, buttons of **Block editor** window’s toolbar and **Block forming** additional toolbar see in the “Project creation” User Manual.

Table 8. Brief description of the Block menu

Menu items	Function
Add images from files	to add image files, locating in the resources out of active profile, to the selected strip (see the “Adding images from files” chapter in “Creating project” User Manual)
Add images from resources	to add image files, locating in the resources of active profile, to the selected strip (see the “Adding images from resources” chapter in “Creating project” User Manual)

Menu items	Function
Add images from resource folder	to automatically add images from specified folder of active profile resources to selected strip, with or without its subfolders
Split to strips	contains menu items used to split images block to strips automatically using images names or exterior orientation parameters (see the “Splitting into strips” chapter in “Creating project” User Manual)
Block editor (**Ctrl+Alt+B**)	to open the **Block editor** window
Block layout (**Ctrl+Alt+L**)	to open the **Block layout** window to build block scheme in 2D-window considering source or acquired data (see the “Aerial triangulation” User Manual)
Create overlap map...	to create map of images and/or strips overlap in new vector layer (see the “Creating overlap map” chapter of the “Creating project” User Manual)
Additional	contains the **Additional** menu (see below)

Table 9. Brief description of the Block menu (Additional)

Menu items	Function
Add strip	to add new strip
Delete strip	to delete selected strips of block in 2D-window
Strip properties	to view and edit properties of selected strips – strip’s name and type (regular or irregular)
Invert image order in strip	to invert strips order
Selected strips to block start	to move selected strips to the beginning
Selected strips to block end	to move selected strips to the end
Add strip	to add new strip
Delete strip	to delete selected strips of block in 2D-window
Strip properties	to view and edit properties of selected strips – strip’s name and type (regular or irregular)
Move selected strips up	to move selected strips one strip up
Move selected strips down	to move selected strips one strip down
Make selected strips irregular	to change regular strip to irregular
Make selected strips regular	to change irregular strip to regular
Delete images	to remove selected images from project
Delete images selectively...	to select images in accordance with specified criteria for deleting (see the “Deleting images selectively” chapter of the “Creating project” User Manual)
Image properties	to display and edit properties of selected image
Move images	opens a group of menu items used to images block editing; allows to invert images order in a strip, and to move selected image left/right/up/down

Menu items	Function
Image radiometric correction	to perform radiometric correction of selected image (see the “Radiometric correction” chapter in the “Project creation” User Manual)
Show images	to show images of block in 2D-window if the limitation on images display is specified on the **Block scheme	Raster** tab in the **Settings** window (see “General information” User Manual)
Show selected images only	to turn on/of selected images display
Check images	to perform search for 16-bit project images without radiometric correction (no `*.rmc` file associated with the image, see “Radiometric correction” in the “General information” User Manual). If images are found in the project that meet the criteria described above, an appropriate info message is issued
Mark all images as checked	to exclude project images validation - check their presence and compliance to images files during project opening at the next working session
Marker to selected image	to move marker to center of the image, selected in the list of the **Block editor** window
Brightness adjustment	to build brightness adjustment for all block scheme (see the “Brightness adjustment” chapter in the “Project creation” User Manual)
Delete brightness adjustment	to delete the brightness adjustment results (see the “Brightness adjustment” chapter in the “Project creation” User Manual)
Rotate selected images	to rotate selected images at any angle relatively to initial or current position of block images
Rotate images by block layout	to rotate all or selected images of block considering block layout data; images don’t rotate if the **w\o layout** was used (see the “Aerial triangulation” User Manual)
Set GSD for selected images	to define/calculate GSD value in meters for all or selected images of block (see the “Specifying ground sample distance (GSD)” chapter in the “Project creation” User Manual)
Create vector layer from block layout	to create vector layers with common block outline, boundaries of all images or selected image of the block (see the “Creating vector layers from block layout” chapter in the “Project creation” User Manual)
Build pre-regions	to create pre-regions by images/stereopairs to provide joint work on a project (see the “Creation pre-regions for a block” chapter in the “Project creation” User Manual)
Export block layout to KML...	to export block scheme to KML format

4.6. The “Orientation” menu

To perform work on data collection for aerial triangulation serves the **Orientation** menu, and also buttons of the main toolbar and of the **AT** additional toolbar.

Table 10. Brief description of the Orientation menu

Menu items	Function
Manage projects cameras (**Ctrl+Alt+I**)	to input/edit parameters of project cameras, and also to assign cameras to project images (see the “Project cameras management” chapter of the “Aerial triangulation” User Manual)
Import orientation from metadata...	to perform import of exterior orientation from metadata (see the “Import orientation from metadata” chapter of the “Aerial triangulation” User Manual)
Import exterior orientation	to open the Exterior orientation data list (see the “Import of exterior orientation parameters” chapter of the “Aerial triangulation” User Manual)
Automatic tie points measurements	contains menu items used to measure tie points coordinates in automatic mode (see the “Automatic measurement of tie points coordinates (general information)” chapter of the “Aerial triangulation” User Manual)
Tie points list (**Ctrl+Alt+T**)	to open catalogue of all *tie points* with their measurements (see “The “Triangulation points” window” chapter of the “Aerial triangulation” User Manual)
GCP list (**Ctrl+Alt+G**)	to open catalogue of *all ground control points*, including non-measured on block images (see the “GCP list” chapter of the “Aerial triangulation” User Manual)
Block adjustment (**Ctrl+Alt+S**)	to open the adjustment module, to view adjustment results and errors correction (see the “Block adjustment” User Manual)
Direct georeferencing	to perform adjustment using imported exterior orientation parameters
Subblock analysis	to perform the subblock analysis by triangulation points (see the “Subblock analysis” chapter of the “Aerial triangulation” User Manual)
Processing report	to create the processing report (see the “Processing report” chapter of the “Aerial triangulation” User Manual)
Delete adjustment results	to delete adjustment data (see the “Block adjustment” User Manual)
Additional	contains the **Additional** menu (see below)

Table 11. Brief description of the Orientation menu (Additional)

Menu items	Function
Report on interior orientation	to open a report about interior orientation results (see the “Report on interior orientation” chapter of the “Aerial triangulation” User Manual)
Manual interior orientation	[only for film camera images] to measure manually coordinates of fiducial marks on images (see the “Manual measurement of fiducial marks coordinates” chapter of the “Aerial triangulation” User Manual)
Semiautomatic interior orientation	[only for film camera images] to perform semiautomatic interior orientation, which is to use image-standard with template of fiducial marks position used to search the same fiducial marks on other project images (see the “Semi-automatic interior orientation” chapter of the “Aerial triangulation” User Manual)
Automatic interior orientation	[only for film camera images] to perform automatic interior orientation, which is the automatic recognition of fiducial marks by their type, peculiar to certain analog camera images (see the “Automatic interior orientation” chapter of the “Aerial triangulation” User Manual)
Calculate interior orientation	to re-calculate interior orientation parameters (see the “Images interior orientation” chapter of the “Aerial triangulation” User Manual)
Delete interior orientation data	to open the **Select images** window that is used to delete results of fiducial marks measurements on selected images (see the “Images interior orientation” chapter of the “Aerial triangulation” User Manual)
Quick Ties	contains menu items used to perform quick ties in order to use the data for block layout creation (see the “Manual images linking (quick ties)” in “Aerial triangulation” User Manual)
Open selected images for measurements	to open the **Points measurement** module used to measure points coordinates in manual mode on images *selected*in 2D-window (see the “” chapter of the “Aerial triangulation” User Manual)
Open images containing marker (**Ctrl+Alt+K**)	to open the **Points measurement** module used to measure points coordinates in manual mode on *images, that contain marker position in 2D-window* (see the “The “Points measurement” module” chapter of the “Aerial triangulation” User Manual)
Open in-strip stereopair	to open the **Points measurement** module used to measure points coordinates in manual mode on selected *in-strip stereopair* in 2D-window (see the “The “Points measurement” module” chapter of the “Aerial triangulation” User Manual)
Open inter-strip stereopair	to open the **Points measurement** module used to measure points coordinates in manual mode on

Menu items	Function
	selected in 2D-window *inter-strip stereopair* (see the “The “Points measurement” module” chapter of the “Aerial triangulation” User Manual)
Report on relative orientation (**Ctrl+Alt+R**)	to open the report with results of images relative orientation in order to perform analysis and removal of errors in tie points measurements (see the “The “Points measurement” module” chapter of the “Aerial triangulation” User Manual)
Delete point measurements	to choose types of triangulation points to be deleted and delete them (see the “The “Triangulation points” window” chapter of the “Aerial triangulation” User Manual)
Clear point catalogue	to remove the whole list of triangulation points (GCP, check, tie) from triangulation points catalogue (see the “The “Triangulation points” window” chapter of the “Aerial triangulation” User Manual)
Load triangulation points	to load measured coordinates of triangulation points to 2D-window with possibility to setup of display of certain type of points (see the “Triangulation points display in 2D-window” chapter of the “Aerial triangulation” User Manual)
Delete points outside useful areas	to delete points out of useful areas (with specified background color) (used mainly during automatic points measurements on spaceborne images)
Delete border points	to delete points of selected types from near-boundary areas of project images marked in the list (see the “Deleting points at image boundaries” chapter of the “Aerial triangulation” User Manual)
Delete duplicated points	to delete tie points duplicates (see the “Removing point duplicates” chapter of the “Aerial triangulation” User Manual)
Import	contains menu items used to import of triangulation points measurements from files of PAT-B, X-Points formats from PHOTOMOD 4.x (XPT) projects, as well as data import from flight path file (see the “Import of triangulation points” chapter of the “Aerial triangulation” User Manual)
Export	contains menu items used to export of triangulation points measurements to files of PAT-B and X-Points formats, and for export of ties or GCP for further use them in *Geomosaic module* (see the “Export of triangulation points” chapter of the “Aerial triangulation” User Manual)
List of exterior orientation parameters	to open the Exterior orientation data list (see the “List of exterior orientation parameters” chapter of the “Aerial triangulation” User Manual)
Export exterior orientation...	to perform export of list of source exterior orientation parameters and adjustment results to PAT-B and CSV formats (see the “Export of exterior orientation

Menu items	Function
	parameters" chapter of the "Aerial triangulation" User Manual)
Load projection centres as vectors	to load projection centres data as vector point objects and to open them in 2D-window to perform analysis, at that an image name is saved to the `Name` point attribute
Create images georeferencing files	to perform export of georeference data after preliminary block exterior orientation and adjustment (to files of *ArcInfo World File*and *MapInfo TAB*formats)
Select subblock	to select part of block images to perform adjustment
Adjustment in batch mode	to specify adjustment parameters and to perform block adjustment without using the the **Adjustment** toolbar (see the "Block adjustment" User Manual)

4.7. The "Terrain" menu

Table 12. Brief description of the Terrain menu

Menu items	Function
Points	The **Points** menu contains usual menu items used for automatic points measurement, filtering, import and export (see the "Points" chapter of the "Creating DTM" User Manual)
TIN	The **TIN** menu contains menu items used to load and save TIN layers, as well as to perform different operations on creation, editing and accuracy control of TIN building
DEM	The **DEM** menu contains standard menu items used to load and save DEM layers, as well as menu items used to perform various operations on DEM creation, accuracy control, filtering and editing
Compute volumes	allows to calculate a volume (i.e. *embankment* or *excavation*) located between DEM surface and **arbitrary** Z-plane or in a more complex case, the volume which is the overlap between the two (roughly, "top" and "bottom") **DEM** or **TIN** surfaces (see the "Volumes calculation" chapter of the "Creating DTM" User Manual)
Contours	The **Contours** menu contains usual menu items used for contours generating, editing, import and export (see the "Contour lines" chapter of the "Creating DTM" User Manual)
LAS	The **LAS** menu contains usual menu items used for converting the LIDAR data to DEM which is saved into the file of active profile (see the "LIDAR Data processing" User Manual)

Menu items	Function
3D-Mod	The **3D-Mod** menu allows to run module for creation 3d-objects based on 2D-vectors (see the "Three-dimensional modeling" User Manual)

4.7.1. The "TIN" menu

The **TIN** menu contains standard menu items used to load and save TIN layers, as well as to perform different operations on creation, editing and accuracy control of TIN building.

The **Points** menu is located in the **Terrain** menu.

Table 13. Brief description of the TIN menu

The TIN menu	Function
Load TIN... (**Ctrl+O, T**)	to load TIN from `*.x-tin` file (see the "TIN loading" chapter in "DTM Generation" User Manual)
Recent	to perform the quick access to recently loaded TIN files
Save	to save or rewrite active TIN layer as file with `*.x-tin` extension (see the "TIN saving" chapter in "DTM Generation" User Manual)
Save as...	to save active TIN layer with new name as file with `*.x-tin` extension (see the "TIN saving" chapter in "DTM Generation" User Manual)
Close	to close TIN
Close all opened layers	to close all opened TIN layers
Layers visibility	contains menu items which allow to perform batch management of TIN layers visibility in the *Layer manager*
 **Build** (**Ctrl+N, T**)	to create TIN using loaded base layers (see the "Building TIN" chapter in "DTM Generation" User Manual)
Build 3D-TIN...	to create textured 3D-TIN surfaces (see the "Creation of textured TIN 3D surface" chapter in "DTM Generation" User Manual)
Build 3D-TIN (Batch mode)	to create textured 3D-TIN surfaces in batch mode (see the "Batch 3D-TIN creation" chapter in "DTM Generation" User Manual)
Convert 3D-TIN CS	to change coordinate system of 3D-TIN (see the "Transformation of 3D-TIN coordinate system" chapter in "DTM Generation" User Manual)
Filter	contains menu items which can be used to filter DEMs (see the "TIN filtering" chapter in "DTM Generation" User Manual)

The TIN menu	Function
Export	Contains menu items to export results of TIN creation to DXF and CSV formats (see the “Export TIN” chapter in “DTM Generation” User Manual)
Import	Contains menu items allowing to import TIN from different formats (see the “Import TIN” chapter in “DTM Generation” User Manual)
Additional	contains the **Additional** menu (see below)

Table 14. Brief description of the TIN menu (additional)

The TIN menu	Function
Restore...	to restore TIN from contours (see the “TIN restoring” chapter in “DTM Generation” User Manual)
Compute border	to create TIN border in automatic mode (see the “TIN borders creation” chapter in “DTM Generation” User Manual)
 **Rebuild**	to rebuild TIN after editing of base layer objects (see the “TIN re-building” chapter in “DTM Generation” User Manual)
Check against adjustment points	to control TIN creation accuracy by triangulation points (see the “Check against adjustment points” chapter in “DTM Generation” User Manual)
Check against vector objects	to control TIN creation accuracy by vector objects, which were not used during TIN creation (see the “Check TIN against vector objects” chapter in “DTM Generation” User Manual)
Verify topology...	to check topology of TIN creation (see the “Verifying of TIN topology” chapter in “DTM Generation” User Manual)
The area of a polygon on the surface	to calculate TIN surface area within a polygon (see the “TIN area info” chapter in “DTM Generation” User Manual)
Calculate area	to calculate TIN projection area on plane and area of TIN 3D surface (see the “TIN area info” chapter in “DTM Generation” User Manual)
Statistic	to view statistic information about TIN (see the “General TIN info” chapter in “DTM Generation” User Manual)
Interpolate	to interpolate TIN to smooth DTM in order to build or enhance contour lines (see the “Smooth TIN interpolation” chapter in “DTM Generation” User Manual)
Convert to vector layer	to convert TIN into a vector layer (see the “Converting TIN into a vector layer” chapter in “DTM Generation” User Manual)
 **Embed objects**	to embed vector objects layer to created TIN (see the “Insert objects into TIN” chapter in “DTM Generation” User Manual)

The TIN menu	Function
Export	Contains menu items to export results of TIN creation to DXF and CSV formats (see the “Export TIN” chapter in “DTM Generation” User Manual)
Import	Contains menu items allowing to import TIN from different formats (see the “Import TIN” chapter in “DTM Generation” User Manual)
On/Off TIN visibility (**Ctrl+T**)	to enable/disable editable TIN layer visibility

4.7.2. The “DEM” menu

The **DEM** menu contains standard menu items used to load and save DEM layers, as well as menu items used to perform various operations on DEM creation, accuracy control, filtering and editing.

The **DEM** menu is located in the **Terrain** menu.

Table 15. Brief description of the DEM menu

Menu items	Function
Load DEM... (**Ctrl+O, D**)	to load DEM from `*.x-dem` file (see the “DEM loading” chapter in “DTM Generation” User Manual)
Open from file...	to load DEM from external data without its conversion to internal format (see the “DEM loading” chapter in “DTM Generation” User Manual)
Recent	to perform the quick access to recently loaded DEM files
Save copy	to save opened DEM to a new file (see the “DEM saving” chapter in “DTM Generation” User Manual)
Save selection	to save DEM area (see the “DEM saving” chapter in “DTM Generation” User Manual)
Close	to close DEM
Close all opened layers	to close all opened DEM layers
Layers visibility	contains menu items which allow to perform batch management of DEM layers visibility in the *Layer manager*
Build DEM...	contains menu items used to create DEM with help of various source data (see the “DEM creation” chapter in “DTM Generation” User Manual)
Filter	contains menu items which can be used to filter DEMs (see the “DEM filtering” chapter in “DTM Generation” User Manual)
Fill null cells	contains menu items used to restore blank cells of DEM using various methods (see the “Null cells in DEM” chapter in “DTM Generation” User Manual)
Import...	contains items used to import DEM from files with `*.grd`, `*.asc`, `*.tif`, `*.dem`, `*.mtw`, `*.dt1`, `*.dt2`,

Menu items	Function
	`*.img`, `*.pix`, `*.hgt` extensions (see the “DEM import” chapter in “DTM Generation” User Manual)
Batch import	to simultaneous import of multiple DEM (see the “Batch DEM import” chapter in “DTM Generation” User Manual)
Batch import from folder	to simultaneous import of multiple DEM from a specified folder (see the “Batch import from folder” chapter in “DTM Generation” User Manual)
Export...	contains menu items used to perform export of DEM to different formats (see the “DEM export” chapter in “DTM Generation” User Manual)
Additional	contains the **Additional** menu (see below)

Table 16. Brief description of the DEM menu (additional)

Menu items	Function
Save as georeferenced raster...	to save a DEM as a raster file with geodetic reference (see the “DEM saving” chapter in “DTM Generation” User Manual)
Build DEM...	contains menu items used to create DEM with help of various source data (see the “DEM creation” chapter in “DTM Generation” User Manual)
Rebuild by TIN	to recreate DEM after changing of TIN base layer (see the “DEM creation by TIN” chapter in “DTM Generation” User Manual)
Restore	is used to restoring of DEM consistency, which may be disrupted by applying various operations or DEM filtering (see the “DEM recovering” chapter in “DTM Generation” User Manual)
Slope based filter	to filter objects based on slope angle (see the “Slope based filter” chapter in “DTM Generation” User Manual)
Filter by image properties	to filter DEMs depending on image characteristics (see the “Filter by image properties” chapter in “DTM Generation” User Manual)
Correct along vector line	to correct DEM cell values along the selected linear objects (see the “Elevation correction along axis line” chapter in “DTM Generation” User Manual)
Cut DEM by Z-range	to convert DEM cells with specified elevation to null cells (see the “Converting cells to null” chapter in “DTM Generation” User Manual)
Set elevation in selected polygons...	to set the same elevation of DEM cells inside and outside of selected polygons (see the “Change DEM parts elevation” chapter in “DTM Generation” User Manual)
Interpolate elevation in selected polygons...	to interpolate a value of DEM cells inside and outside of selected polygons (see the “DEM interpolating” chapter in “DTM Generation” User Manual)

Menu items	Function
Cut by selected polygons	to edit coverage area of DEM (see the “DEM cutting by polygons” chapter in “DTM Generation” User Manual)
Cut null edges	to remove edge areas of DEM, that include blank cells (see the “DEM cutting by polygons” chapter in “DTM Generation” User Manual)
Transpose	to transpose DEM from the left coordinate system to the right one and vice versa (see the “DEM transpose” chapter in “DTM Generation” User Manual)
Transform to another coordinate system	to change coordinate system of DEM (see the “Transformation of DEM coordinate system” chapter in “DTM Generation” User Manual)
Batch transform to another coordinate system	to change coordinate system of multiple DEMs (see the “Batch transformation of DEMs coordinate system” chapter in “DTM Generation” User Manual)
Rebuild considering last adjustment	to recreate DEM considering results of the last adjustment (if the project was adjusted after DEM creation once more – see the “Rebuilding DEM considering last adjustment” chapter in “DTM Generation” User Manual)
Shift	to parallel displacement of DEM by axes (see the “DEM shift” chapter in “DTM Generation” User Manual)
Split DEM into sheets	to split DEM into sheet to save DEM in parts in separate files (see the “Split DEM into sheets” chapter in “DTM Generation” User Manual)
Merge DEMs	to merge DEMs, that have overlap area (see the “DEM merging” chapter in “DTM Generation” User Manual)
Accuracy control	to perform accuracy control DEM creation using various data (see the “Accuracy control of DEM creation” chapter in “DTM Generation” User Manual)
Compute volumes	to calculate a volume (i.e. *embankment* or *excavation*) located between DEM surface and **arbitrary** Z-plane or in a more complex case, the volume which is the overlap between the two (roughly, “top” and “bottom”) DEM surfaces (see the “Volumes Calculation” chapter in “DTM Generation” User Manual)
Calculate slope map	to calculate a slope map (see the “Slopes map creation” chapter in “DTM Generation” User Manual)
Convert to points...	to convert DEM cells to points with specified simplifying (see the “Convert DEM to points” chapter in “DTM Generation” User Manual)
Convert to LAS	to convert DEM to LAS point cloud (see the “Convert DEM to point cloud (LAS)” chapter in “DTM Generation” User Manual)

Menu items	Function
Write points to DEM	to add points to DEM that helps to refine DEM cells values (see the “Adding of points to DEM” chapter in “DTM Generation” User Manual)

4.7.3. The “Contours” menu

The **Contours** menu contains usual menu items used for contours generating, editing, import and export.

The **Contours** menu is located in the **Terrain** menu.

Table 17. Brief description of the Contours menu

Menu items	Function
Load... (**Ctrl+O**, **V**)	to load contour lines from `*.x-data` file (see the “Vectorization” User Manual)
Recent	to perform the quick access to recently loaded contour lines
Save	to save or rewrite active contours layer (see the “Saving contour lines” chapter in “DTM Generation” User Manual)
Save as...	to save active layer with new name (see the “Saving contour lines” chapter in “DTM Generation” User Manual)
Save selected as...	to save *just* selected contours (see the “Saving contour lines” chapter in “DTM Generation” User Manual)
Close	to close contour lines layer
Build contours	contains menu items used to create contour lines with help of various source data (see the “Contour lines generation” chapter in “DTM Generation” User Manual)
Import	contains items used to import contour lines from files with different extensions (see the “*Import of vector objects*” chapter of the “Vectorization” User Manual)
Export	contains menu items used to perform export of contour lines to files with different extensions (see the “*Export of vector objects*” chapter of the “Vectorization” User Manual)
Additional	contains the **Additional** menu (see below)

Table 18. Brief description of the Contours menu (additional)

Menu items	Function
Build contours	contains menu items used to create contour lines with help of various source data (see the “Contour

Menu items	Function
	lines generation" chapter in "DTM Generation" User Manual)
Check contours intersections...	to check created contours for crossing/self-crossing, that occur after operation of contours smoothing (see the "Control of contours crossing" chapter in "DTM Generation" User Manual)
Check contours by points	to perform quality control of contours creation using regular points, if they were not used for contours creation (see the "Check contours by points" chapter in "DTM Generation" User Manual)
Merge contours...	to merge created contours in automatic or manual mode (see the "Contours merging" chapter in "DTM Generation" User Manual)
Check merging contours...	to check contours for breaks that occur after manual or automatic contours merging (see the "Check of contours merging quality" chapter in "DTM Generation" User Manual)
Contours connecting	to connect created contours automatically or manually (without merging into a single vector object – see the "Contours connecting" chapter in "DTM Generation" User Manual)
Check contours vertices	to check elevations of created contours vertices (see the "Check elevations of contours vertices" chapter in "DTM Generation" User Manual)
Precision of objects coordinates	to set a precision of vector objects coordinates at level of decimal places (see the "Precision of objects coordinates" chapter in "DTM Generation" User Manual)

4.8. The "Vectors" menu

See the "Vectorization" User Manual.

4.9. The "Raster" menu

The system provides the **Raster** menu to perform various operations with raster images.

Table 19. Brief description of Raster menu

Menu items	Function
Orthorectification	to create orthophoto production and mosaicking (see the "Orthorectification" User Manual)
GeoMosaic	to run the *GeoMosaic program* allows to create the orthomosaic from georeferenced orthorectified aerial and satellite imagery, splitting of created orthomosaic with the capable of saving sheets in popular raster file format (see the "Orthophotomaps creation" User Manual)

Menu items	Function
Raster Converter	to launch the *Raster Converter* program for preparing and converting raster images into the internal format files and then for placing them in active profile resources (see the "General information" User Manual).
ImageWizard module	to edit path to image files (see the "ImageWizard Adjustment of images" chapter in "Creating project" User Manual)
Load georeferenced images (files)	to load georeference images from folder in file system (see the "Georeferenced external data" section of the "Aerial triangulation" User Manual and the "Pseudo-stereo mode" section of the "Vectorization" User Manual)
Load georeferenced images (resources)	to load georeference images from active profile resources (see the "Georeferenced external data" section of the "Aerial triangulation" User Manual and the "Pseudo-stereo mode" section of the "Vectorization" User Manual)
Load web-map	to load georeferenced web-maps from the Internet (see the "Web-map loading" chapter in "Creating project" User Manual)
Show rasters	to use one of three modes of displaying raster images in 2D-window: **Cached only** (**Ctrl+Shift+1**), **Depending on zoom** (**Ctrl+Shift+2**) or **Source only** (**Ctrl+Shift+3**)
Additional	contains the **Additional** menu (see below)

Table 20. Brief description of Raster menu (additional)

Menu items	Function
Image georeferencing	to perform georeferencing images by geodetic coordinates GCP points, obtained from vector/raster maps or from a list of file `*.txt` (see the "Orthophotomaps creation" User Manual)
Affine georeference correction	to perform affine correction of georeferenced images (see the "Affine georeference correction" chapter in "Creating project" User Manual)
Save raster layer	to save a georeference raster image of the active layer with specified parameters in output file format (TIFF, MS-TIFF, MegaTIFF – see the "Saving of raster image" chapter in "Creating project" User Manual)
Close all opened layers	allows to close all opened raster layers
Layers visibility	contains menu items which allow to perform batch management of raster layers visibility in the *Layer manager*
ScanCorrect	to run the *ScanCorrect* program for compensation of metric errors occurred when scanning graphical

Menu items	Function
	data on flatbed polygraphic scanners (see the "ScanCorrect program" User Manual)
Dust Correct	to 'clean' images of dust particles, photo emulsion defects etc (see the "Dust Correct" chapter in "Creating project" User Manual)
Clear cache	to delete raster cache from RAM
Images order	to rearrange block images by z-order
Rebuild Mega Tiff pyramids	to rebuild image pyramids for MegaTIFF images (see the "Image pyramid creation" chapter in "Creating project" User Manual)
Adjust zoom	to set zoom of images in 2D-window by set GSD in pixels

4.10. "Service" menu

Table 21. Brief description of the Service menu

Menu	Function
Explorer	to open the *PHOTOMOD Explorer* module to view resources system (see the "PHOTOMOD Explorer module. Resources management" chapter in "General information" User Manual)
GeoCalculator	to launch the GeoCalculator program that is used for coordinates transformation form one reference system to another
Show in Google Maps	to display area on images in the GoogleMaps service (Internet connection required); it is possible only for projects in geodetic coordinate systems or if the coordinate system could be transformed to geodetic
Show in Yandex Maps	to display area on images in the YandexMaps service (Internet connection required); it is possible only for projects in geodetic coordinate systems or if the coordinate system could be transformed to geodetic
Recalc working area	to refresh 2D-window and fit displayed area depending on load data
Activate mouse driver	to turn on/off defined mouse driver
Mouse setup	to setup or connect mice, including special mice, hand wheels and foot pedals, and also to macros setup (see the "Using of special mice, hand wheels and foot pedals in the system" chapter in "General information" User Manual)
Undo (**Ctrl+Z**)	to cancel the last operation of vector objects editing on a layer (see the "Undo editing operations" chapter in "Vectorization" User Manual)

Menu	Function
Undo log	to open the **Undo log** containing a list of recent editing operations (see the “Undo editing operations” chapter in “Vectorization” User Manual)
Redo (**Ctrl+Shift+Z**)	to redo the last undone operation (see the “Undo editing operations” chapter in “Vectorization” User Manual)
Last log	to display the last log of system actions
Parameters	to open the window to set the general parameters of the system
Additional	contains the **Additional** menu (see below)

Table 22. Brief description of the Service menu (additional)

Menu	Function
Distributed Processing	contains menu items to run, test and setup distributed processing (see the “Distributed Processing” chapter in “General information” User Manual)
Working coordinate system	to choose working coordinate system (see the Project creation User Manual)
Autodetect Gauss-Krueger zone	to detect the Gauss-Krueger zone automatically for the selected object, provided that any Global coordinate system is used (see the “Automatic detection of the Gauss-Krueger zone” chapter in “General information” User Manual)
CSV converter	to launch the CSV converter to transform points coordinates in CSV and TXT files and also for other transformations for CSV files (see the “CSV converter” chapter in “General information” User Manual)
Load atlas	to load the World map to a new vector or raster layer (see the “Loading atlas” chapter in “General information” User Manual)
Save scene	to save visible part of images in active 2D-window as a raster image with specified size and quality
Customize hotkeys	to adjust hotkeys using in the system, edit, delete or create new hotkeys (see the “Customize hotkeys” chapter in “General information” User Manual)
Customize fast commads	to use so called shortcut commands, the custom button combinations for quick access to various functions when working with the layers such as **Vectors**, **DEM**, **Raster**, **Grid**, or **TIN** (see the “Shortcut commands” chapter in “General information” User Manual)
Save options	to save projects parameters and use it automatically when restart the system

4.11. The “Window” menu

Table 23. Brief description of the Window menu

Menu items	Function
New 2D-window (block)	to open window with a block scheme
New 2D-window (stereopair) (**Ctrl+Alt+W**)	to open window with a stereopair
3D-window	to open the 3D-window (see the “General information” User Manual)
Marker window (**Ctrl+Alt+C**)	to open marker parameters window (see the “’Vectorization” User Manual)
Measurements window (**Ctrl+Alt+D**)	to open window that allows to perform measurements by images (see the “Vectorization” User Manual)
Classifier	to open the **Classifier** window to show set of standard vector objects attributes (see the “Classifier $QUOTE$" chapter in the “Vectorization” User Manual)
Objects attributes	to open the **Attributes** window to display attributes of selected vector objects (see the “Vector objects attributes $QUOTE$" chapter in the “Vectorization” User Manual)
Toolbars	contains menu items allows to open additional toolbars
Additional	contains the **Additional** menu (see below)

See detailed description of the system’s windows in the “General information about the system” User Manual.

To get quick access to the *main* windows of the system, select **Window › Toolbars › Windows**. The **Windows** toolbar is opened.

Table 24. Brief description of the Window menu (Additional)

Menu items	Function
Block editor	to open block editor window (see the ’Project creation’ User Manual)
Block layout	to open the block layout window (see the ’Aerial triangulation’ User Manual)
Show all toolbars	to show all toolbars
Find all toolbars	to show all opened toolbars in visible part of the screen
Temporarily hide toolbars (**TAB**)	to hide/show all opened toolbars
Image list...	to open the **Images list** window (see the ’Creating project’ User Manual)
New 2D-window (single image)	to open window with image selected on a block scheme

Menu items	Function
Open image under marker	to open all 2D-windows with marker place. Press and hold **Alt** key while clicking the menu item to open images with 1:1 zoom , otherwise, full images are displayed
Close all single image windows	to close all 2D-windows with images
Refresh all 2D-windows	to refresh information in all opened 2D-windows
Refresh all 2D-windows	to close all 2D-windows
Arrange	contains menu items allow to arrange opened 2D-windows in a work area with one of the following ways: • **Tile** – to tile 2D-windows; • **Row** – to row 2D-windows; • **Column** – to column 2D-windows; • **Stack** – to stack 2D-windows; • **Arrange 2D-windows in a tabsheet**; • **Evenly** – to arrange 2D-windows evenly.
Stereopairs	contains menu items allows to move to other stereopairs (see the 'Vectorization' User Manual)
Triangulation points coordinates	to show table with triangulation points coordinates
Triangulation points view control	to choose triangulation points to display
Contours classifier table	to open contours classifier window to edit parameters of contours display (see the 'DTM generation' User Manual)
Objects list	to display list of active layer vector objects (see the 'Vectorization' User Manual)
Marker motion in pixel coordinates	to set on the mode allows to move marker in stereopair 2D-window in pixel coordinates
Zoom all in (**Shift+* NumPad**)	to zoom in all 2D-windows
Zoom all out (**Shift+/ NumPad**)	to zoom out all 2D-windows
Layer view mask	to use a view mask for active layer (see the 'General information' User Manual)
Windows list	to view list of opened 2D-windows

Appendix A. Preparing to work

A.1. Program start

Prior to work in the program, it is needed to insert *Sentinel HL* unique security key into the USB-socket of the workstation. The security key drivers should be installed on the

workstation (see “Protection of the system” chapter of “General information” User Manual).

To start the program do the following:

1. Choose **Start › Science › PHOTOMOD UAS 8.0**;

2. [optional] Prior to work in the program you should create a *profile* for placing project resources such as project configuration files, images files, processing results.

   Profile creation is performed during the first launch of *PHOTOMOD UAS* program or in the *Control Panel* module (see the “Control Panel. Profiles management” chapter of the “General information” User Manual).

3. Wait until the *System Monitor* module starts. The icon in the system tray appears. Wait until *PHOTOMOD UAS* main window opens.

   If the *System Monitor* module is already launched – choose **PHOTOMOD UAS** item in the context menu of this module (the icon in the system tray), to open previously closed *PHOTOMOD UAS* user interface.

4. The **Project management** window opens in *PHOTOMOD UAS* user interface:

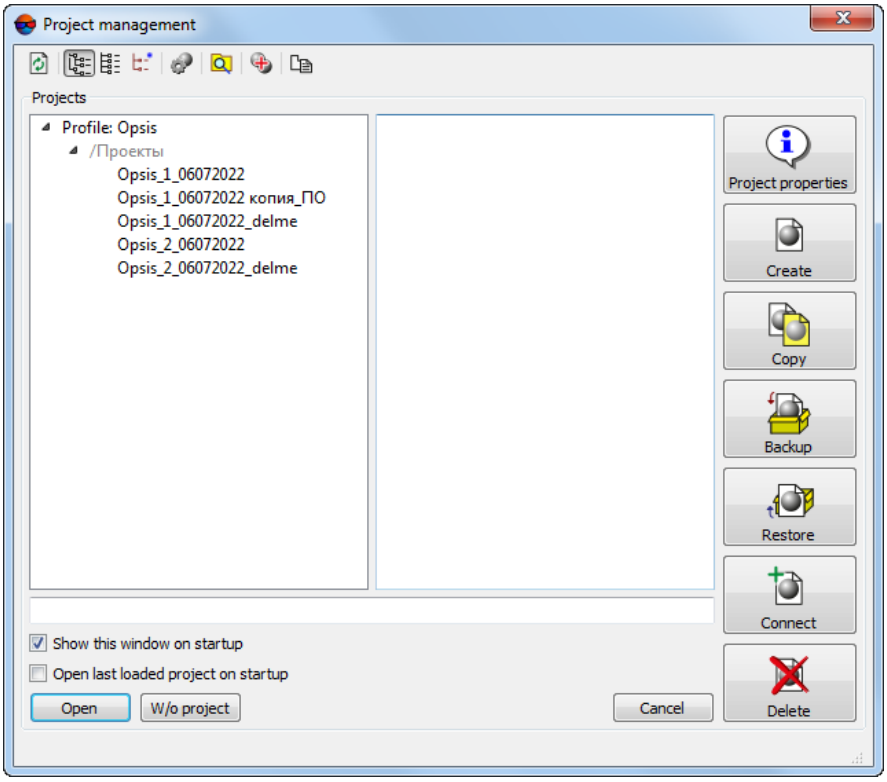


Fig. A.1. The Project management window

To close the **Project management** window click **W/o project** button.

The **New project** window also opens automatically if there is no existed projects in the active profile. Click **Cancel** to close it.

5. The main *PHOTOMOD UAS* window in ready for processing:

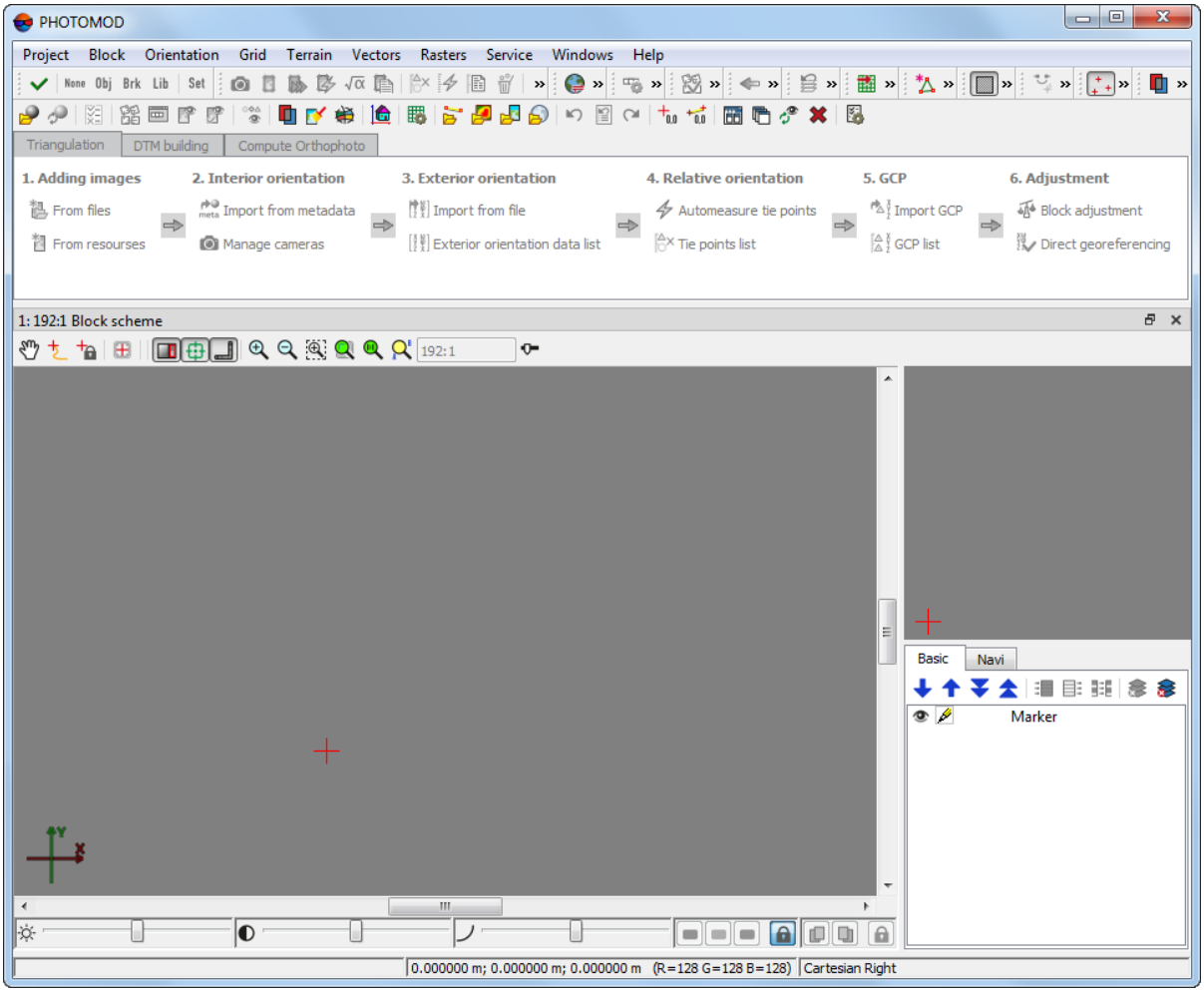


Fig. A.2. The main PHOTOMOD UAS window (without project)

A.1.1. Choosing active profile

This step is optional.

During one work session of the program could be used **only one** *active profile*. User can change the active profile before project creation, if *PHOTOMOD UAS* software is already configured and connected to existing profiles and resource systems.

A.1.1.1. Brief description of data storage organization

See detailed description in “Prepare to processing” chapter of the “General information” User Manual).

Resources system is a way to store data with possibility of network distributed project processing. Operator has access for an unified system with all available profiles, regardless of the PC used for data storage and processing.

The following main definitions are used in the resources system:

- *Profile* – independent group of resources relative to one or several projects;
- *Virtual folders* is a virtual names of real local/network folders (hard disks) or group of folders, chosen by user to store profile resources.

One profile can use data located on several computers.

Configuration of resources depends on profile settings. Profile has a virtual name and do not equal to real file system. This name is a common root of the resources tree that connects all branches (resources) profile. The tree have the following structure:

- *Root* – profile virtual name.
- *Top level branches* – profile virtual folders – virtual names of physical local or network folders/disks.

One physical folder could be specified as a virtual folder.

- *Resources* – the whole content of selected folder of a profile – all subfolders and files. Only images source files from active profile resources could be matched with images of current project.

Profiles can be *local* or *network* to organize individual or shared work with projects, respectively. Configuration files that describe a set of local profiles and its structure are stored in the *PHOTOMODUAS8.VAR* folder.

For profiles creation and organizing resources structure of each profile you can use the *Control Panel module*. To view and edit active profile resources is used the *PHOTOMOD Explorer module*.

To launch these modules is used the **Start › Science › PHOTOMOD UAS 8.0** menu and also the context menu of the *System Monitor* module (the  icon in the system tray).

A.1.1.2. Changing active profile

Close all program modules (except *System Monitor*) and perform one of the following actions to change the *active profile*:

- choose active profile in the **Profile** list in the context menu of the *System Monitor* module (the icon in the system tray);
- choose the **Control Panel** menu item in the context menu of the *System Monitor* module. The *Control Panel* module opens. Select the active profile with mouse double-click by profile name in the **Profile** list in the *Control Panel* module window.

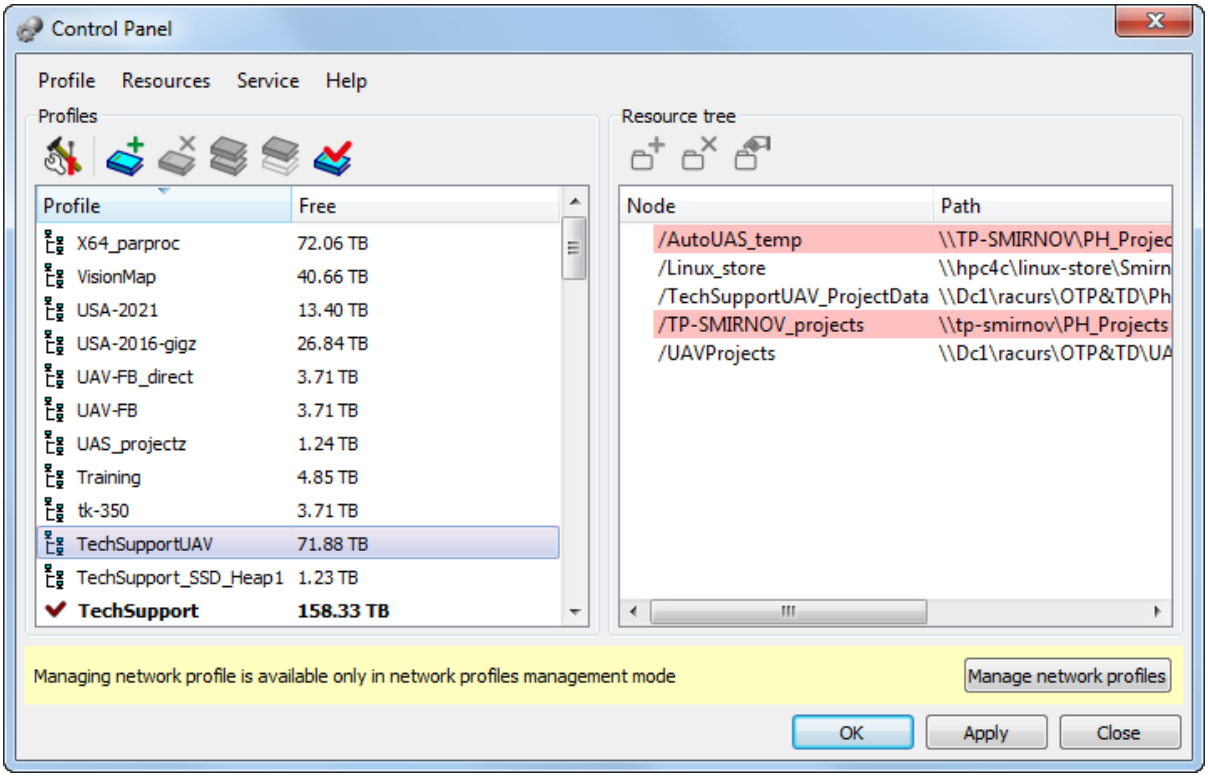


Fig. A.3. The Control Panel module window

Current active profile is marked in the profile list by the icon.

Restart all program modules after changing the active profile (to do this, choose **Exit** item in the context menu of *System Monitor* module, close all opened program windows and run *PHOTOMOD UAS* again). All resources in the active profile are available for use until another active profile is selected.

A.2. Image preparing

Data preparation, i.e. converting images into the internal *PHOTOMOD* format and loading them into the internal resource system is an optional (and recommended) stage of data processing. The system provides for loading unprepared images into the project directly from the file system. However, the further description of project processing implies processing of prepared images loaded into the active profile resource system.

Prior project creation it is recommended to prepare UAS images using the *Raster Converter* module.

Raster Converter is a service system module that could be used to batch and sequence conversion of raster files to inner MS-TIFF format with LZW or JPEG conversation or without it and to convert MS-TIFF images to different formats.

For working with images in program, it is recommended to use MS-TIFF internal raster format, which is the TIFF format with overview pyramid (set of subsampled images copies) for higher image display performance.

The system allows the direct use of various formats of raster images (without having to convert them). It saves the disk space, but slows down significantly operations with images, so it is strongly recommended to use raster images in MS-TIFF format.

In case of processing amount of images places on different data medium and take much disk space, it is recommended to preliminary prepare raster images with module before creation of a project.

To start the module perform one of the following:

- choose **Start › Science › PHOTOMOD UAS 8.0 › Raster Converter**;
- choose **Rasters › Raster Converter**;
- choose **Raster Converter** in the right-click menu of the *System Monitor module* (the  icon in the system tray).

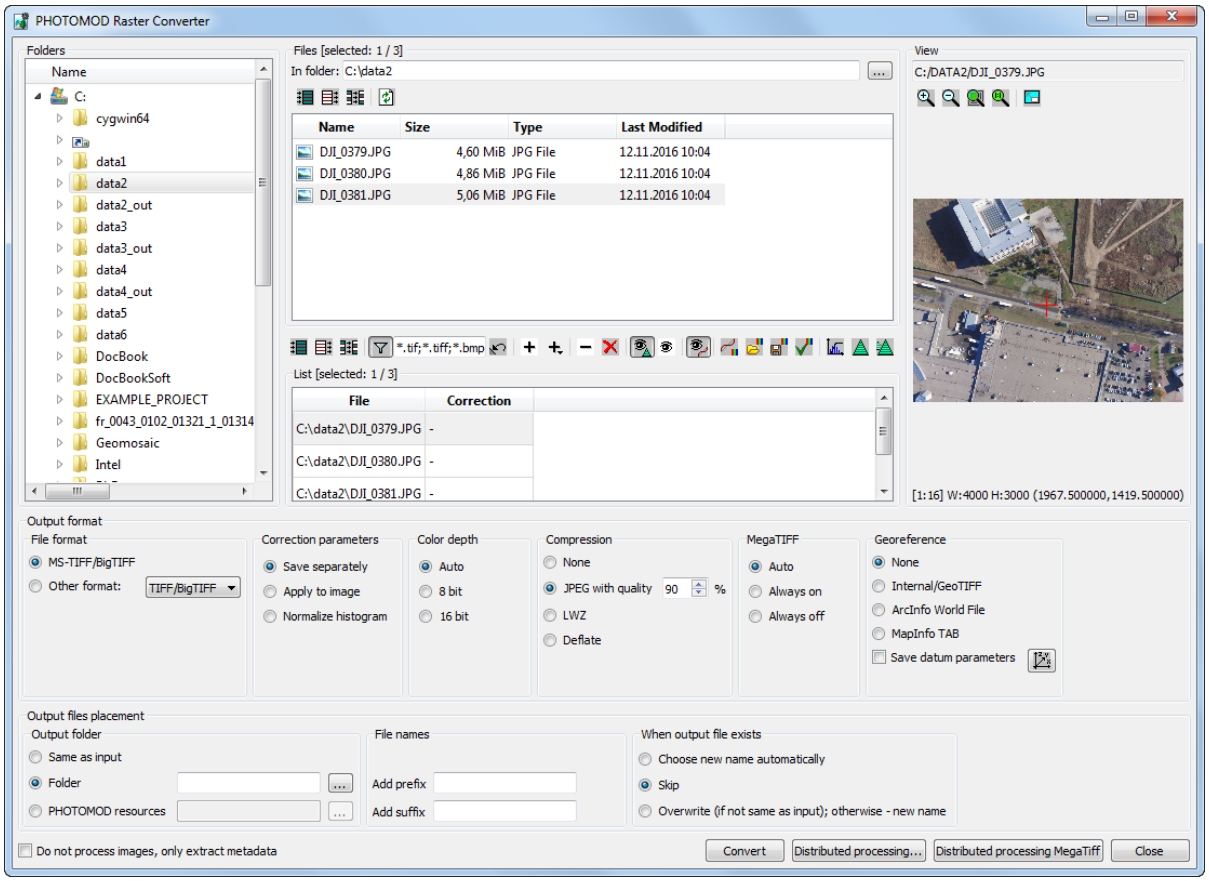


Fig. A.4. Raster converter window

The recommended workflow for the central projection project images:

See the full *Raster Converter* module description in the "General information" User Manual.

1. [if necessary] Perform the radiometric correction;
2. Convert the images to files of inner *MS-TIFF* format;
3. [optional] It is recommended to apply **JPEG compression** with **85 % quality** for output files to save place on hard disk;
4. Choose an individual folder in **PHOTOMOD resources** as the **output folder** for the converted images.

Further, when creating a project (after converted images have been saved), do not place it in the folder with images. The most optimal solution is to place the project and the converted images in two different folders in the resource system.

Appendix B. UAS project processing sequence

PHOTOMOD UAS is intended to work exclusively with **Central projection** project types (see the “Project types” section of the “Creating project” User Manual). The project processing process may vary depending on the features of the input data and the required results.

Processing of *PHOTOMOD UAS* project implies performing the following main steps: **Aerial Triangulation**, **Compute DEM** and **Compute Orthophoto**. The appropriate main toolbar tabs names, layout and content displays the main steps of UAS data processing workflow (see Section 4.2.1).

Key *PHOTOMOD UAS* functionalities similar to corresponding *PHOTOMOD* instruments (aerial triangulation, vectorization, DEM creation, and orthophoto) are described in the appropriate sections in manuals from the main *PHOTOMOD* documentation package (see Section 4.3).

Possible procedure options are shown below as diagrams. A step-by-step example of project processing is given in Appendix C.

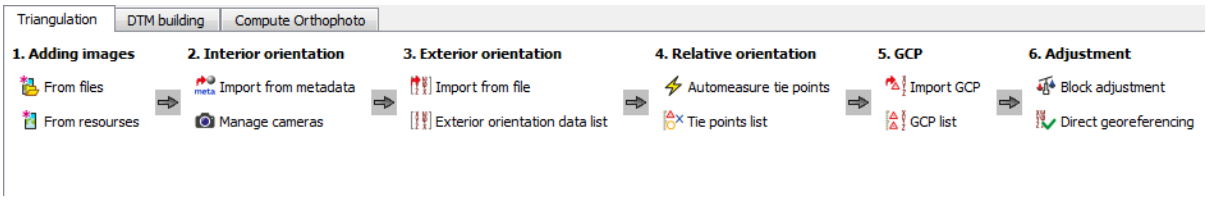


Fig. B.1. UAS project processing sequence (Triangulation)

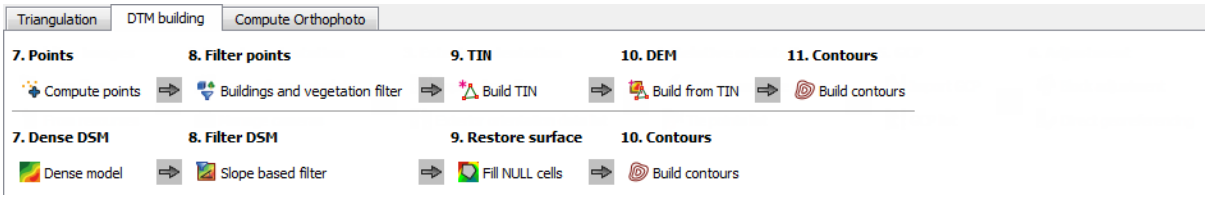


Fig. B.2. UAS project processing sequence (DTM building)

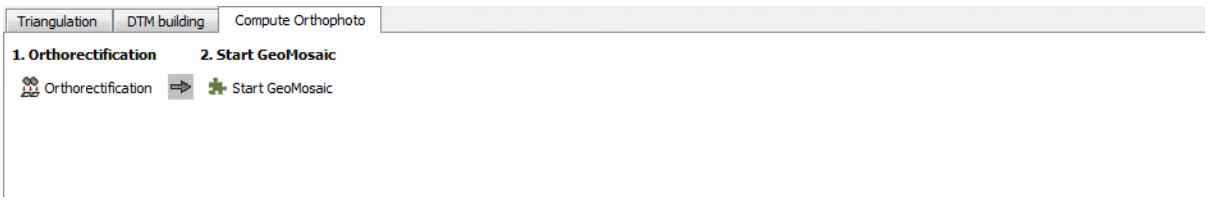


Fig. B.3. UAS project processing sequence (Compute orthophoto)

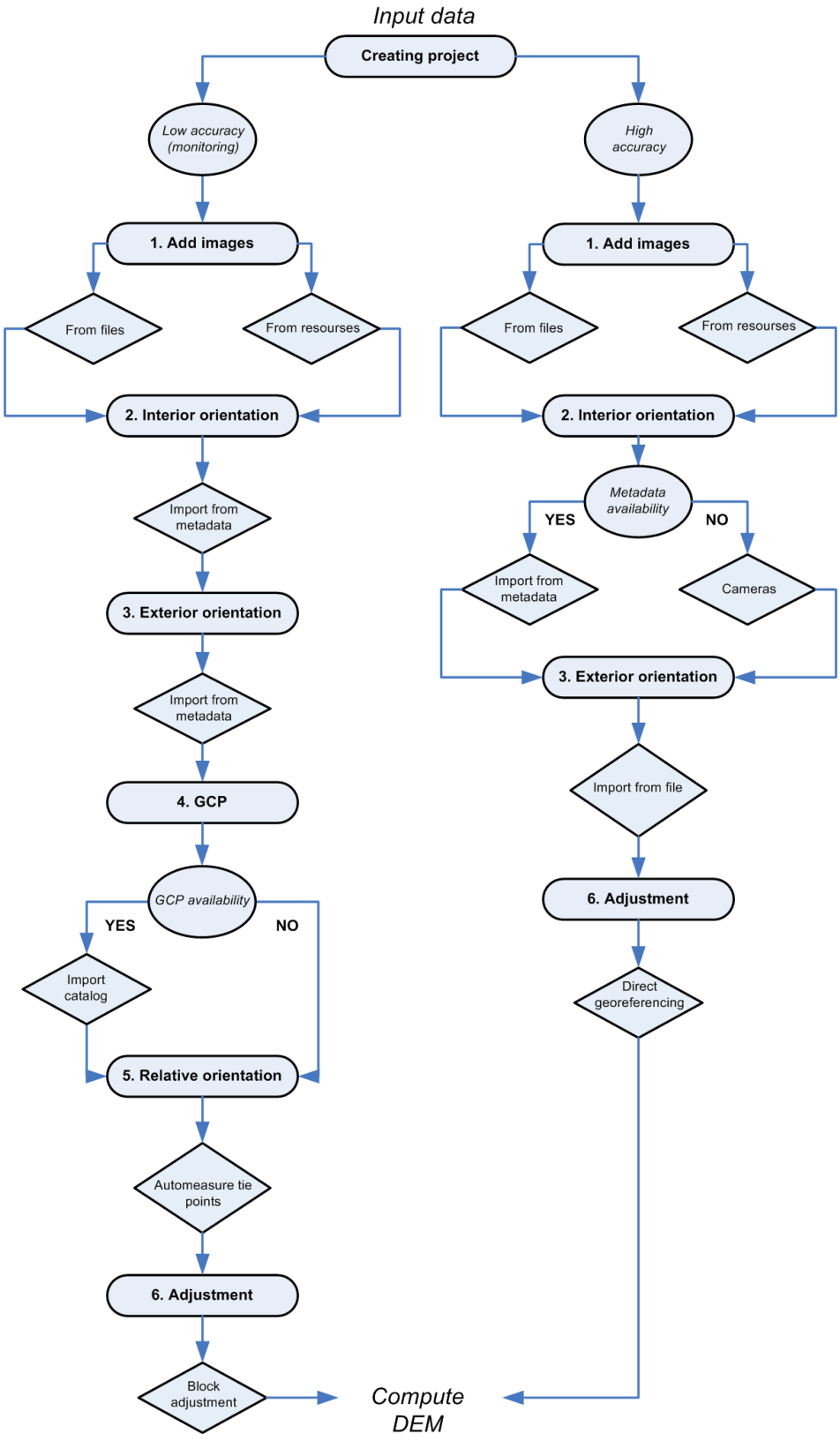


Fig. B.4. Presumed workflow of UAS processing

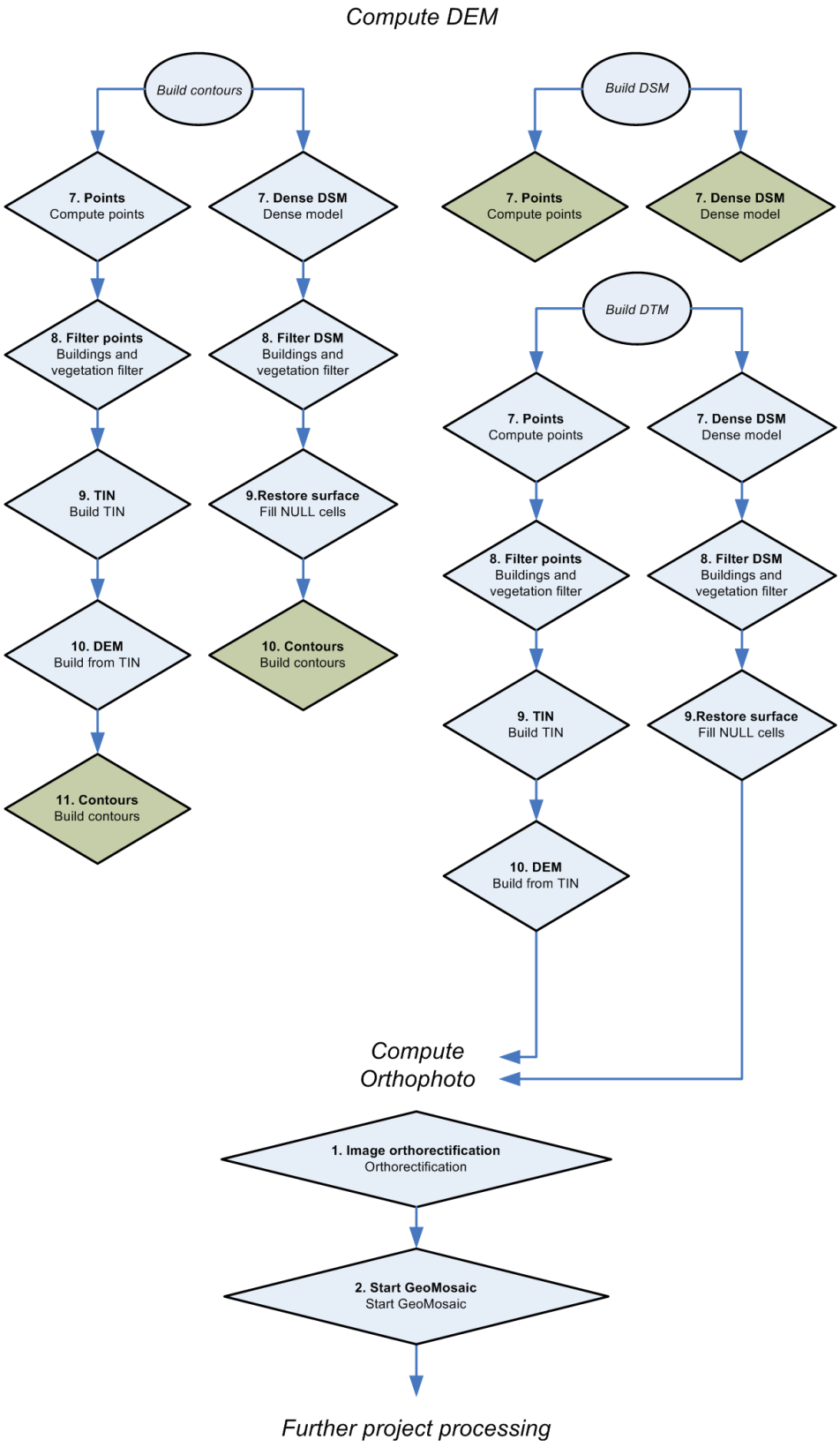


Fig. B.5. Presumed workflow of UAS processing

Appendix C. UAS project processing example

This section discusses the most common algorithm for processing a UAS project. More detailed user manual, including those that consider non-standard situations that arise during data processing, are contained in the main *PHOTOMOD* documentation.

C.1. Project creation and images adding

See details about central projection project creation in the “Project creation” User Manual.

In order to create UAS project perform the following actions:

1. Start the *PHOTOMOD UAS* program;
2. Open the **New project** window using one of the following ways:
   - choose **Project › Open/Manage...** (**Ctrl+Alt+O**). The **Manage project** window opens. Click the **New** button;
   - choose **Project › New**.

     The **New project** window opens automatically if there is no existed projects in the active profile.

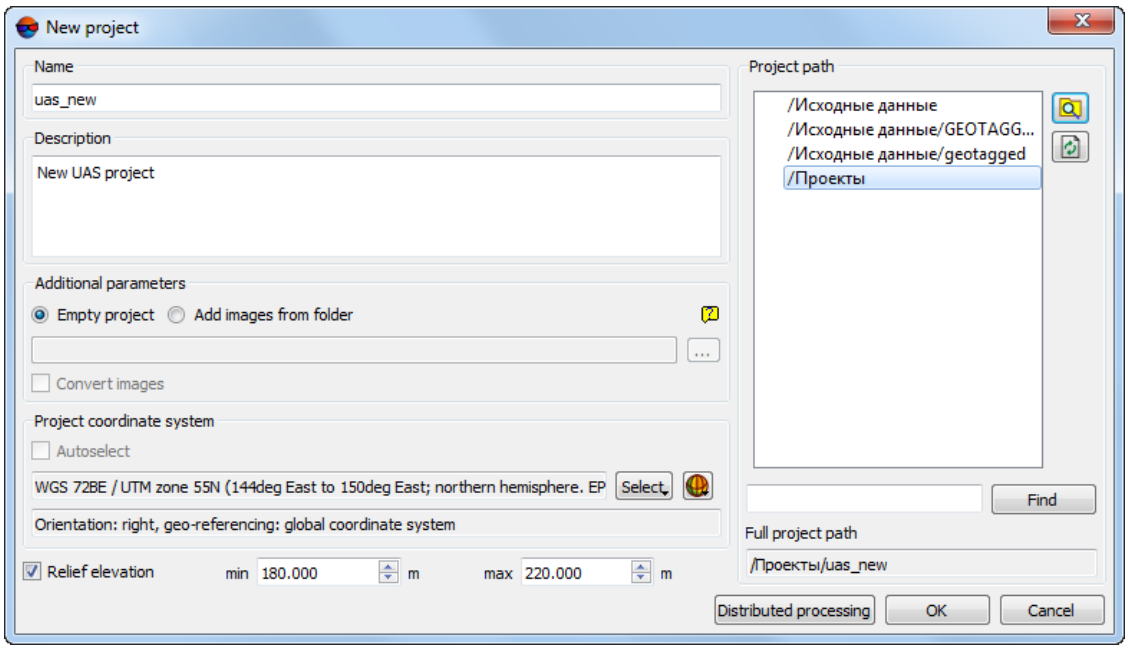


Fig. C.1. Creating UAS project.

3. Define a **Name** of new project.

4. [optional] Input to the **Description** section a brief project description, its main features, notes etc.

5. In the **Coordinate system** section click the **Select** button. Project coordinate system could be chosen from the following sources:

   - **From DB**: **International**, **Local** – used for coordinate system choosing from international or local database.

   - **From file** – used to load a coordinate system from the `*.x-ref-system` file (if any), placed outside from active profile resources.

   - **From resource** – used to load a coordinate system from the *.x-ref-system file, located in active profile resources (from another project of active profile, for instance);

   - **From GeoCalculator** – from the list of *GeoCalculator* program database (see the “Coordinate Systems” chapter in “The GeoCalculator program” User manual).

   The *GeoCalculator* program allows to edit coordinate systems, to create new ones, to perform import and export of coordinate systems (see the “The GeoCalculator program” User Manual).

When choosing coordinate system from database the **Coordinate system database** opens, which contains the list of coordinate systems.

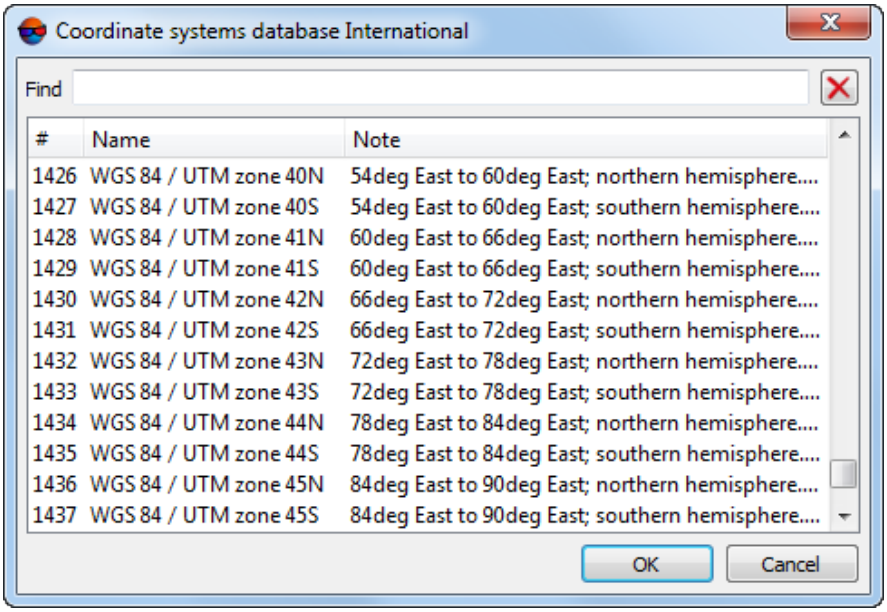


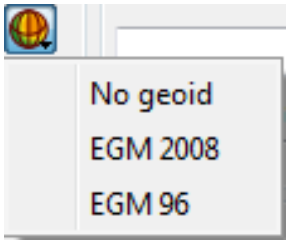
Fig. C.2. Window used to select coordinate system from coordinate system database

To perform fast search for coordinate system, input the whole coordinate system name or its part to the **Find** input filed.

To choose geoid to be used, click the button. Select proper type of geoid usage:

- **No geoid**;
- **EGM 96**.

The system allows to use the **EGM2008** geoid. See more details in the "Installation EGM2008 Geoid" User Manual. After installation the geoid is displayed in the list.



PHOTOMOD GeoCalculator provides for creating a user height system (geoid) with preset parameters. *PHOTOMOD* system allows to use previously created custom vertical datum (geoid). See instructions in the "Creating custom height system" chapter of "The GeoCalculator program" User Manual.

6. [optional] Set the **Relief elevation** checkbox and specify approximate difference of terrain heights on project images in the **min** and **max** input fields.

   If the relief elevation data is unavailable or inaccurate by the time of project creation, the system allows to define the value later in project properties. Terrain elevation data is used to refine a block layout, it is considered during import of exterior orientation parameters and during calculation of ground sample distance (GSD).

   If the **Relief elevation** was not indicated while project creation, it is in any case will be set when importing exterior orientation elements (see the "Aerial triangulation" User Manual).

7. In the **Project path** list choose a folder of active profile resources to place project files there.

   Only folders of two upper levels could be used to place project files into the **Project path** list (virtual folders and their subfolders), which do not contain images files. To edit structure of profile resources use the *PHOTOMOD Explorer* module (see the "General information" User Manual).

   Virtual folders connection to network profile is performed only in the*Control Panel module*. After update of profile resources structure click the button to refresh the list in the **Project path** section.

   The search field that allows to **Find** a project in the list by its name or name part is situated below the **Project path** list. **F3** key allows to focus marker on the search field. Press **F3** again to focus marker on the next found project in the list.

   The **Full project path** input field will show the project name and full path.

8. Click OK to complete a project creation. After that service folders and configuration files are created in the specified project folder. The first strip create automatically (see below).

   It is possible to use the distributed processing mode during project creation. For this, perform the following:

   1. Change settings and run the distributed processing server/client (see the “Distributed processing” chapter in the “General information about system” User Manual).

   2. Click the **Distributed processing** button.

9. The **New strip** window opens:

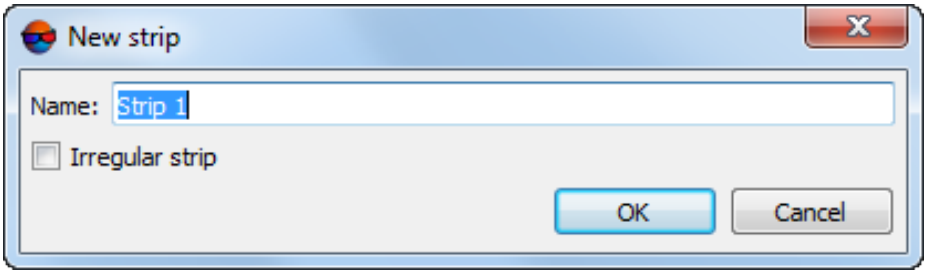


Fig. C.3. The New strip window

10. Define **Name** of the strip.

11. Click OK. New strip creates and its frame displays in block scheme 2D-window. The **Block editor** window opens (see below). The row with name of strip displays in the table of the **Block editor** window.

C.1.1. Adding images from resources

Refer to the “Images block forming” chapter of “Project creation” User Manual for the detailed description of images loading from *files* and from *resources*, images setup and images block editing.

Perform the following actions to load prepared images to a project from the active profile resources and to form an images block:

1. Click **From resources** button in **Triangulation** tab of the main program toolbar (or click the button in **Block editor** window toolbar).

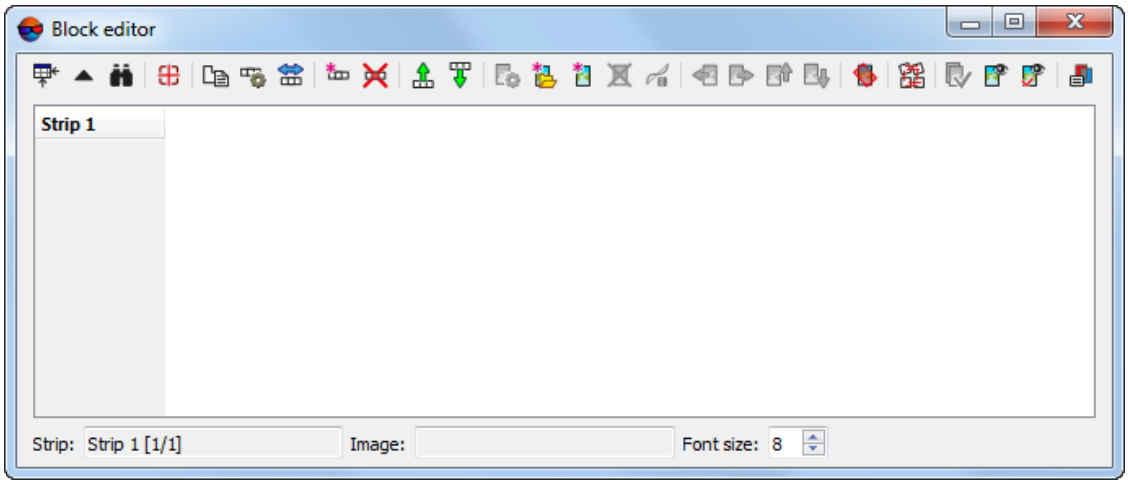


Fig. C.4. The Block editor window

2. Choose prepared images files and click the **Open** button. After that all images are loaded to the first strip.

To filter showing files choose `Image in TIFF format` in the list rightward to the **Resources name** field.

To add images from the file system – click **From files** button in **Triangulation** tab of the main program toolbar (or choose **Block › Add images from files**).

C.2. Interior orientation

This section *briefly* describes the process of interior orientation of images obtained by a *digital* camera. It covers importing interior orientation data from metadata (if available) and setting up camera parameters (if it is not possible to import data).

The detailed description of the process of interior orientation of images (including images obtained by an analog camera) see in the “Interior orientation” section of the “Aerial triangulation” User Manual.

Interior orientation of image acquired by *digital camera* is to determine the principal point position relative to the center of the lower left pixel of the image. Interior orientation operation includes input or import of camera parameters, specifying of flight direction and angle of camera axes rotation for images of a project.

C.2.1. Import from metadata

In most cases, source images acquired by digital camera, *might* contain metadata, written in EXIF format. Metadata of converted source images (in TIF format or in system’s internal format MS-TIFF) is stored in the folder, which contains these images, in separate files with `*.md` extension.

In both cases, if images containing metadata are available in a project, it is possible to import from the metadata to the project parameters of interior orientation (precise) and exterior orientation (approximate, i.e. calculated by on-board devices). In order to perform interior orientation when metadata is available, perform the following actions:

Prior to orientation data import from images metadata it is recommended to specify height difference of terrain in the project in the **Project properties** window (see the "Project creation" User Manual).

1. Choose **Orientation › Import exterior orientation** or click the **Import from metadata** button on the **Triangulation** tab of the main porgram toolbar to get camera data from EXIF-metadata in image files. The **Import orientation from metadata** window opens:

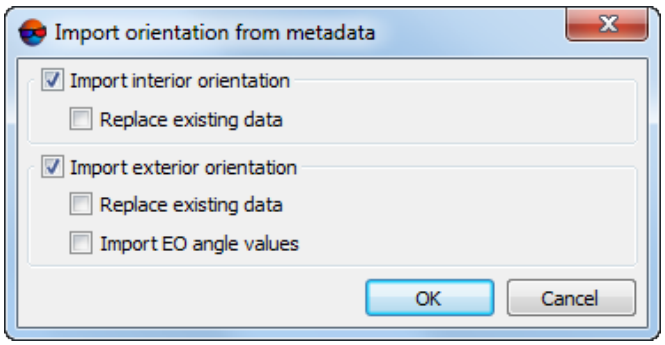


Fig. C.5. Import orientation parameters from EXIF-metadata

2. Make sure that the **Import interior orientation** checkbox is set on.

   The **Import preliminary exterior orientation** checkbox is also set by default, in order to import *exterior* orientation parameters.

   Exterior orientation parameters imported from images metadata allow to split a project on strips using imported exterior orientation, rotate images by block layout using imported data, adjust block scheme by imported exterior orientation parameters.

   Set the **Import EO angle values**, if needed.

   Angular exterior orientation parameters of image – parameters that define image angular orientation in coordinate system of photogrammetric survey object, slope and rotation angles of image.

   To perform import of exterior orientation parameters it is necessary that a project is in coordinate system consistent with WGS-84 coordinate system.

3. [optional] Set **Replace existing data** checkboxes, if the information about internal and external orientation has always been imported into the system and data currently imported is more preferable.

4. Click OK.

[optional] To check the results of exterior orientation parameters import, perform the following actions:

1. Choose **Orientation › Manage cameras** or click the **Manage cameras** button on the **Triangulation** tab of the main program toolbar to adjust camera parameters and define camera axes direction for project images. The **Manage project cameras** window opens.

2. Choose camera in the list and click the button. The **Camera** window opens to view and edit camera data:

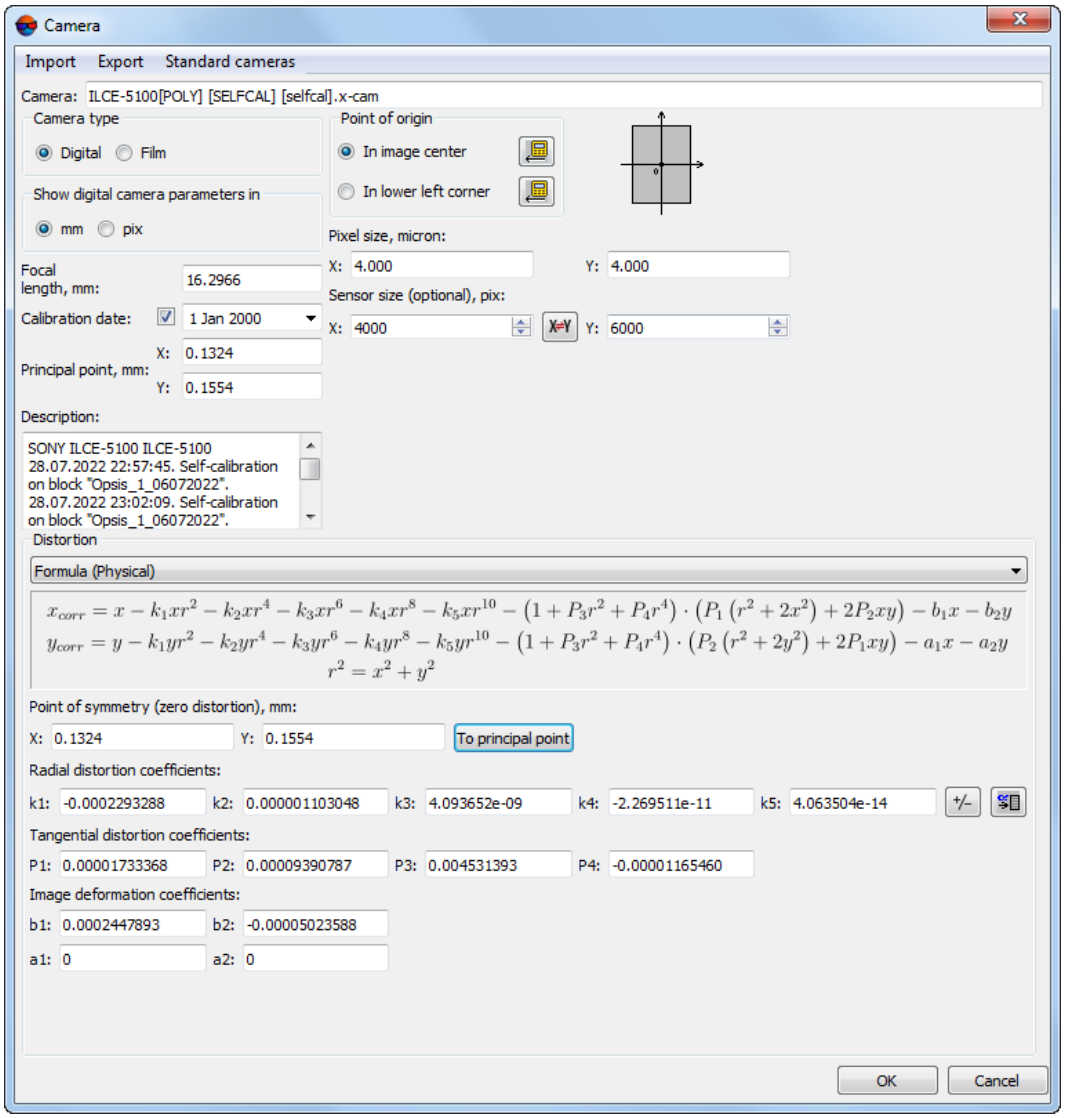


Fig. C.6. The Camera window containing imported data

C.2.2. Manage cameras

In order to perform interior orientation when metadata is unavailable, perform the following actions:

1. Choose **Orientation › Manage cameras** or click the button of main toolbar to adjust camera parameters and define camera axes direction for project images. The **Manage project cameras** window opens.

2. Choose camera in the list and click the button. The **Camera** window opens to view and edit camera data.

3. [optional] Input camera additional data (if camera passport is available) and click OK.

4. Specify flight direction using window of images preview in **Manage project cameras** window and define camera axes rotation for project images in the following way:

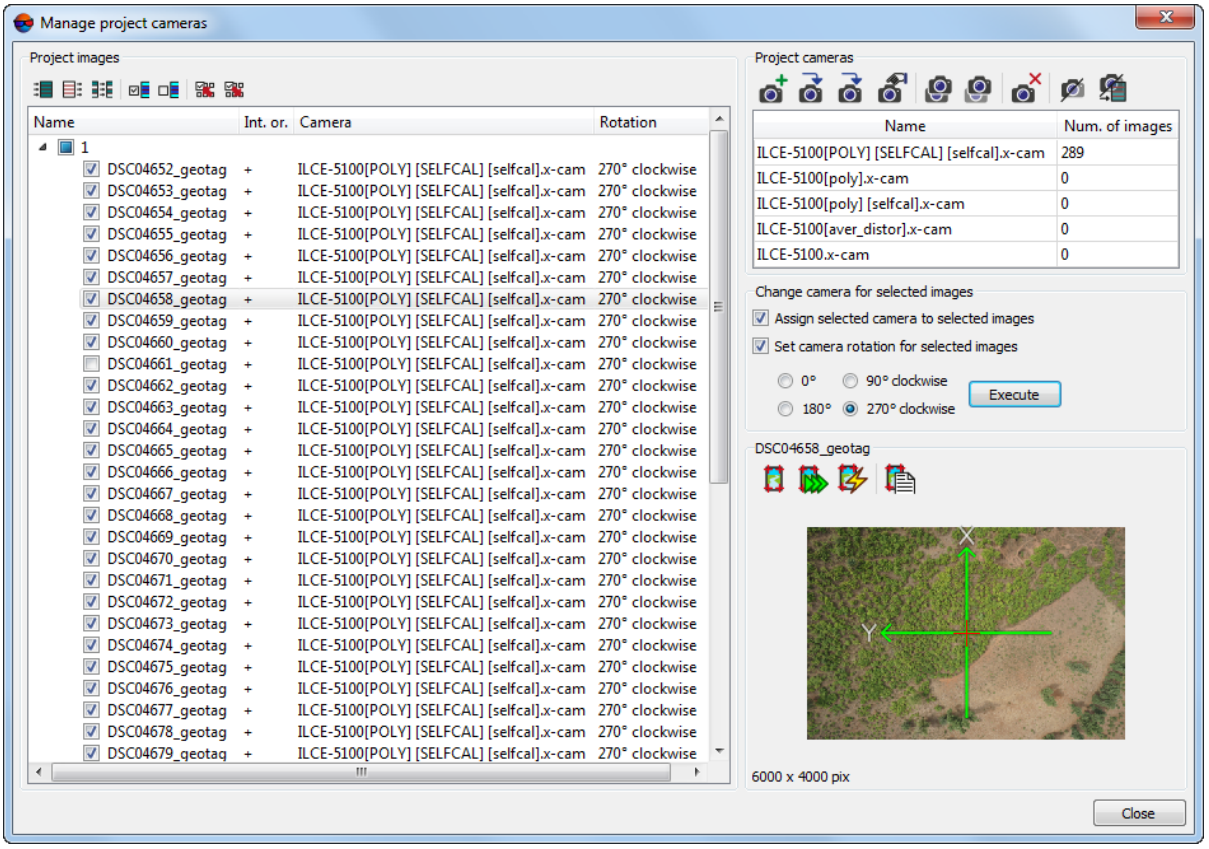


Fig. C.7. The Manage project cameras window

1. Choose images in the table which do not require axes rotation.

2. Set the **Set camera rotation for selected images** checkbox and specify a rotation angle so, that X axis coincides with flight direction.

3. Click the **Execute** button. Interior orientation is performed automatically.

See detailed description of this interior orientation step in the “Project cameras management” chapter of “Aerial triangulation” User Manual.

C.3. Exterior orientation

This user manual addresses the process of importing exterior orientation elements of images from image metadata or from separate text files. For a detailed description of the exterior orientation of images, see the “Exterior orientation” section of the “Aerial triangulation” User Manual. A separate chapter below describes working with ground control points.

Exterior orientation parameters are *coordinates of projection centres and 3 angles*, that allow to specify in whole real position of images in space.

Exterior orientation parameters may be *included to EXIF-metadata of images files* or *separately as a catalogue in a file of text format*. Additional information about import of preliminary exterior orientation data from EXIF-metadata is in Section C.2.

If exterior orientation parameters are available for UAS project, you can import them to the project, together with Interior orientation parameters.

C.3.1. Import from file

In order to perform import of exterior orientation data catalogue from a file of text format, do the following:

1. Choose **Orientation › Additional › Exterior orientation data list** or click the **Exterior orientation data list** button on the **Aerial Triangulation** tab of the main toolbar. The **Exterior orientation parameters** window opens.

2. Click the button to import exterior orientation parameters.

   The button is duplicated by the **Import from file** button on the **Triangulation** tab of the program main toolbar and the **Orientation › Import exterior orientation...** menu item.

3. Choose a file with exterior orientation parameters and click OK. The **Exterior orientation import – Step 1 of 3: File** window opens. The **File name** input field displays selected file placement and name.

   To select another file which contains exterior orientation parameters, click the ... button.

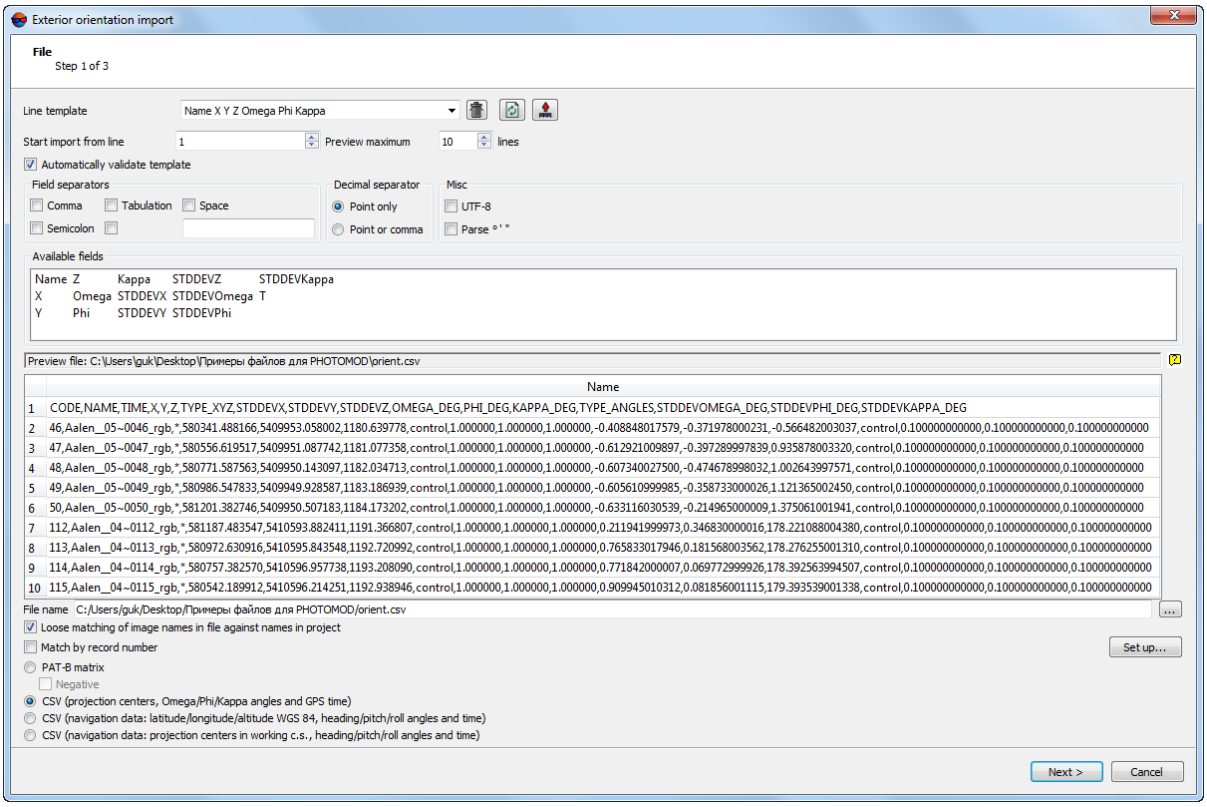


Fig. C.8. The Exterior orientation import window (the file string template is not specified)

If the imported data contains information about time stamps, this information must be presented in *Unix time* (*POSIX time*) format.

Unix time is represented by an integer that increments with each passing second. The integer system used is convenient for comparing and storing dates.

If one needs to access date elements (day, month, year), seconds can be converted into any suitable format (and vice versa). A signed integer type is used in programs to store *Unix time*.

4. [optional] If images names in file and in project are not the same, perform one of the following actions:

   - set on the **Loose matching of image names in file against names in project** checkbox. The program performs search for common substrings and matches images names. For example, it searches for common substring *018_02595* in image name *RGBI_018_02595* in file and image name *018_02595* in project.

     If the **Loose matching of image names in file against names in project** checkbox is set on the image name with file extension is also imported.

   - set on the **Match by record number** checkbox and click the **Setup...** button. The **Match images by numbers** window opens.

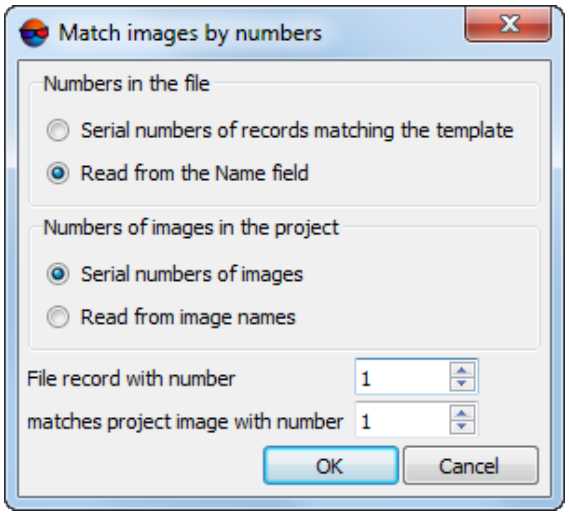


Fig. C.9. Images matching by numbers

1. In the **Numbers in the file** section set the following parameters:

   - **Serial numbers of records matching the template** – is used to match a list of records numbers in a file by order;
   - **Read from the Name field** – is used to match a list of records numbers in a file by the last number from the filename.

2. In the **Numbers of images in the project** section set the following parameters:

   - **Serial numbers of images** – used to match a list of records numbers in a project by order;
   - **Read from images names** – used to match a list of records numbers in a project by the last number from the filename.

3. Input necessary parameters to the **File record with the number** and **matches project image with number** input fields.

   For convenient setup of necessary parameters it is recommended to place windows in that way that allows to display on screen a name of the first image in the table of the **Exterior orientation parameters** window, and also allows to display first rows in the **Preview file** table of the **Exterior orientation import** window. It is recommended to place the **Match images by numbers** window in such a way that allow to show both windows mentioned above.

5. Choose one of the following formats to specify a type of imported file:

   - **PAT-B matrix** – file containing coordinates of projection centers and rotation matrices.

If the PAT-B file contains exterior orientation parameters, calculated for negative plane, set the **Negative** checkbox.

In case of data export from the **PAT-B matrix**, the further line template setup is not required. Click **Next** to move to the second step.

- **CSV (projection centers, Omega/Phi/Kappa angles and GPS-time)** – file containing coordinates of projection centers in any coordinate system, as well as exterior orientation parameters, and precise time from GPS-receiver.

  Line template by default: `Name`, `X`, `Y`, `Z`, `Omega`, `Phi`, `Kappa` (see below).

  CSV format is exchange text format with `*.csv` extension, which is supported by major applications in different industries. It is used as exchange format when special geospatial data formats are not applicable for some reason.

- **CSV (navigation data: latitude/longitude/altitude WGS-84, heading/pitch/roll angles and time)** – file containing navigation data (latitude/longitude set in degrees, altitude in meters, angles in degrees, radians or gons) and precise time from GPS-receiver.

  Line template by default: `Name`, `Lat`, `Lon`, `H`, `Heading`, `Pitch`, `Roll`.

  To perform import of exterior orientation parameters it is necessary that a project is in coordinate system consistent with WGS-84 coordinate system.

- **CSV (navigation data: projection centers in working c.s., heading/pitch/roll angles and time)** – file containing navigation data (coordinates of projection centers in project coordinate system, angles in degrees, radians or gons) and precise time from GPS-receiver.

  Line template by default: `Name`, `X`, `Y`, `Z`, `Heading`, `Pitch`, `Roll`.

6. [optional] If data is imported from `*.csv` format, it is necessary to setup a current line template.

   The **Line template** field displays the list of main and additional fields, contained in each line of imported file (depending from selected import format):

   - `Name` – image name;
   - `X`, `Y`, `Z` (or `Lat`, `Lat`, `H`) – coordinates of center of optical image projection;
   - `Omega`, `Phi`, `Kappa` (or `Heading`, `Pitch`, `Roll`) – Omega, Phi, Kappa angles (or flight heading, roll and pitch);

- [optional] STDDEVX, STDDEVY, STDDEVZ – accuracy of *linear exterior orientation parameters* (RMS);
- [optional] STDDEVOmega, STDDEVPhi, STDDEVKappa – accuracy of *angular exterior orientation parameters* (RMS);
- T – time from GPS device;
- * – fields, that will be missed during import.

The **Preview file** table contains data of imported file. Fields type according to the template, located in the **Line template** field, are automatically assigned to the table columns.

All objects are saved using the same template. Each line of a file contains the same number of fields, that equals to number of fields in template. Lines which does not correspond to the template, are skipped.

In order to begin the setup of active template, perform the following:

- [optional] To display necessary number of lines in the **Preview file** table, set the **Preview maximum** parameter. Default number of lines is 10.
- [optional] To specify a line from to start data import, define the value of the **Start import from line** parameter.

In many cases, the imported file may have a first row containing headers describing the contents of the columns in the file. Accordingly, this row should not be included in the imported data, but its display in the **Preview file** table may be useful when setting up the row template. It is recommended to set this parameter at the very end, when proceeding to the next step.

- In the **Field separators** section set on one or multiple checkboxes to specify possible fields delimiter symbol: **Comma**, **Space**, **Tab**, **Semicolon** or **Other** delimiters. Default settings are comma and space.
- In the **Decimal separator** section setup the following parameters:
  - **Point only** – to use point only as a decimal separator in coordinates;
  - **Point or comma** – to use both point and comma as a decimal separator in coordinates.

If comma is used as **Field separator**, it is not recommended to use comma as **Decimal separator**, since it may cause import of incorrect data.

- [optional] In the **Misc** section set the checkbox:

○ **UTF-8** – is used to recognize text in *Unicode* encoding;

Unicode – symbols encoding format that allows to provide symbols of almost all written languages.

○ **Parse ° ’ “** – is used to recognize records of projection centers or GCP.

When using this parameter it is highly recommended to check points coordinates recognition correctness after import.

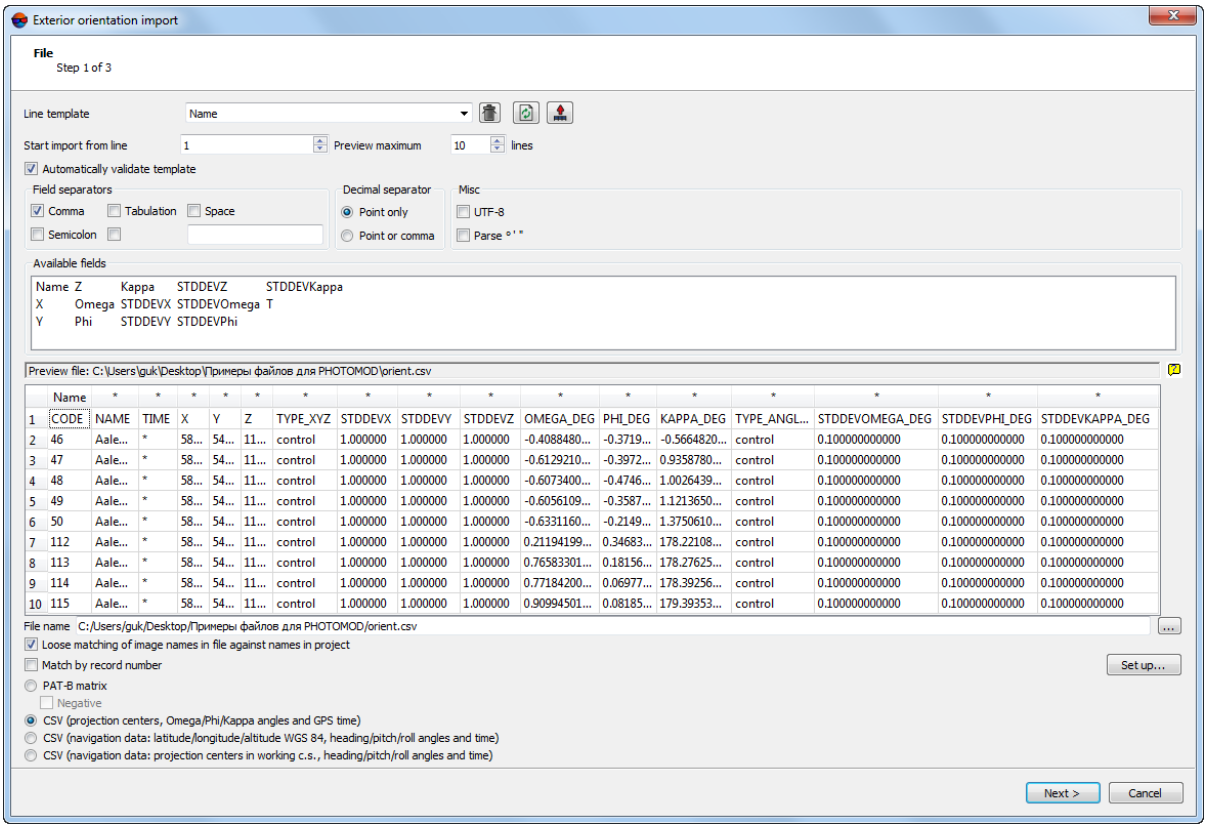


Fig. C.10. Setup a file string template for import of exterior orientation parameters (the table is divided into columns, but their order is not correct)

In order to finish the setup of the active template, perform one of the following actions:

- drag by mouse field name from the **Available fields** list to the head of the column of the **Preview file** table. After that the template in the **Line template** field changes. In order to cancel field selection, double click the column of the **Preview file** table column;
- change the template manually in the **Line template** field. At that column types in the **Preview file** table are changed automatically.

To cancel the field name double click the column.

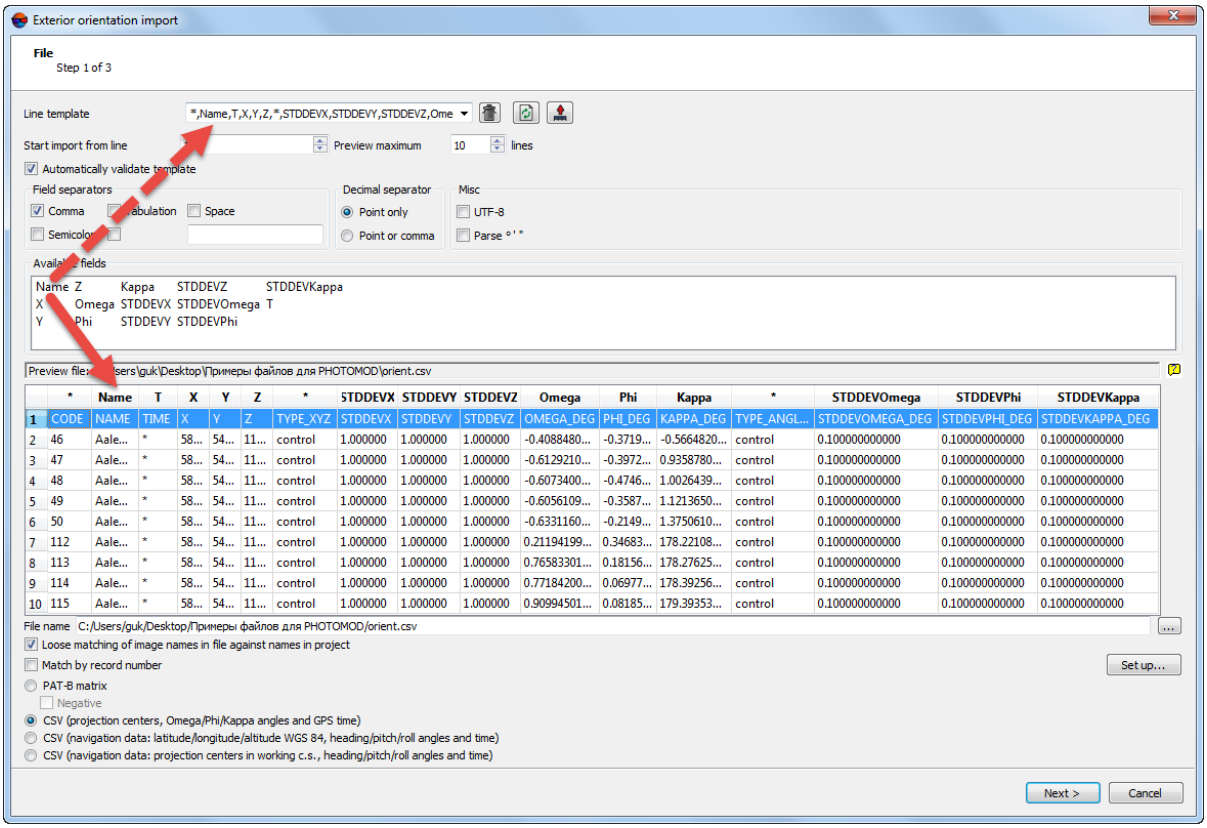


Fig. C.11. Setting up the file line template to import exterior orientation parameters (correct data entered in the Line template field). The first line of the file that needs to be excluded from import is highlighted, starting the import from line 2

The button allows to use the default template.

The button is used to compare the **Line template** field with data shown in the **Preview file** table.

The button is used to replace specified field names by field values taken from the first line of the **Line template** table.

[optional] The **Automatically validate template** checkbox is set on by default that allows to select current template automatically. In order to configure the template for file containing lines with different columns number, set off the **Automatically validate template** checkbox and configure the template in the **Line template** field manually.

7. Click the **Next** button. The **Exterior orientation import – Step 2 of 3: Import settings** window opens.

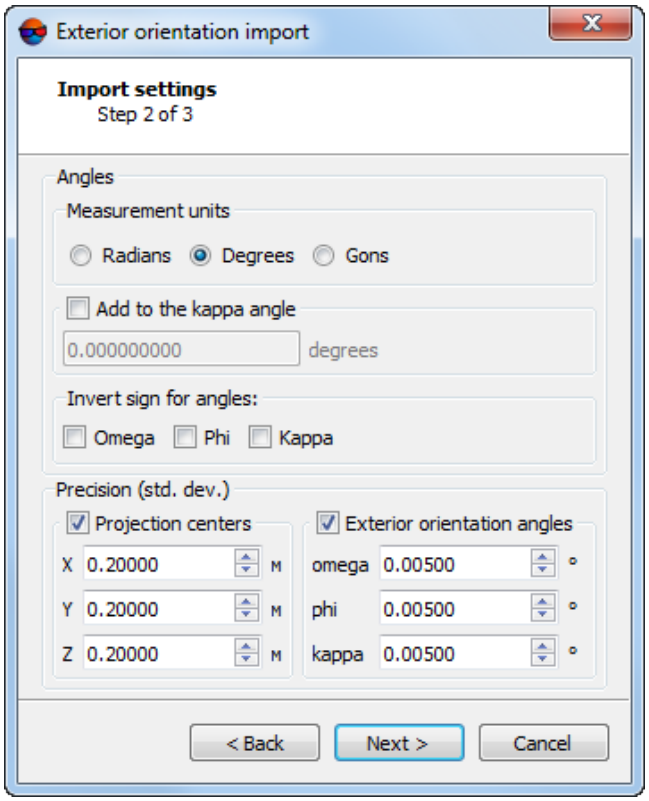


Fig. C.12. Import options of exterior orientation parameters

Define the following parameters of import:

- In the **Angles** section specify measurement units for angles coordinates according to data of the file to be imported:
  - **radians**;
  - **degrees**;
  - **gons** – plane angle measurement unit that equals to 1/100 of flat right angle measurement unit, and full angle is 400 gons.
- [optional] In order to make correction to kappa angle, set on the **Add to the kappa angle** checkbox and input correction value to the input field.
- [optional] In order to **Invert sign** for angles **Omega**, **Phi**, **Kappa** (or **Heading**, Pitch, **Roll**), set on corresponding checkboxes.
- [optional] In the **Precision (std. dev.)** section change precision of a priori RMS of measurements of projection centers coordinates and exterior orientation angles.

If **Projection centers** and **Exterior orientation angles** checkboxes are cleared, the appropriate data are imported from the file (if the data are available and the import parameters are correctly configured in the **Exterior orientation import – Step 1 of 3:**

File window). Imported data may differ from image to image. If the checkboxes are set, the values manually specified by a user are used during import (the same for all images).

8. If import of exterior orientation parameters is performed using the **Exterior orientation parameters** window, then click the **Execute** button. After that the exterior orientation parameters are imported to the **list of exterior orientation parameters**.

   If import of exterior orientation parameters is performed using the **Orientation › Exterior orientation import...** menu item or the **Import from file** button on the **Aerial Triangulation** tab of the main toolbar, then click the **Next** button. The **Exterior orientation import – Step 3 of 3: Additional actions** window opens.

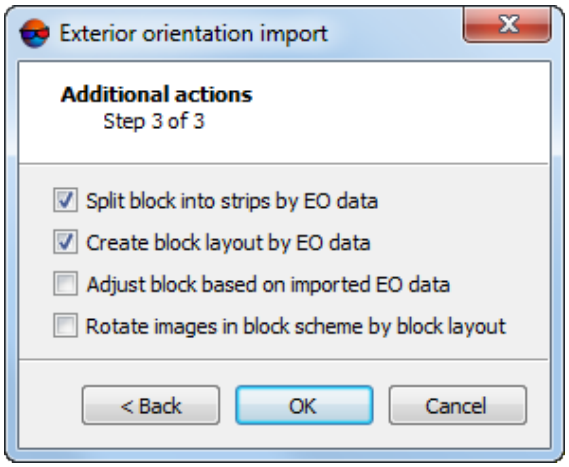


Fig. C.13. Additional actions after exterior orientation import

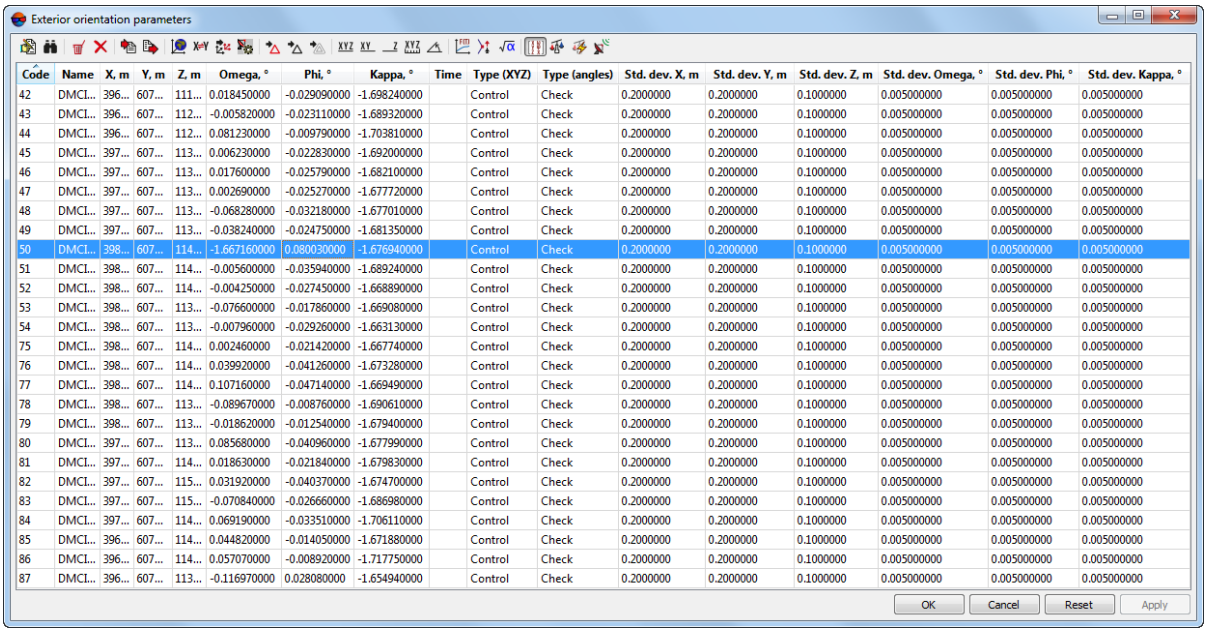
   Select and set the following checkboxes to perform automatic execution of these actions just after import of exterior orientation parameters from file:

   - **Split block into strips by EO data** – to split block images into strips using imported exterior orientation parameters (if all images were loaded to a single strip on a step of project creation);
   - **Create block layout by EO data** – to create block layout considering imported exterior orientation parameters;
   - **Adjust block based on imported EO data** – to perform block adjustment using imported exterior orientation parameters;
   - **Rotate images in block scheme by block layout** – to re-create a block scheme in 2D-window using block layout по накидному монтажу (this checkbox is available if the **Create block layout by EO data** is set).

9. Click the **Execute** button. Exterior orientation parameters obtained after the import are displayed in the table of the **Exterior orientation parameters** window.

10. Click the **Apply** button in the **Exterior orientation parameters** window to save exterior orientation parameters folder in the project.

11. Click OK. The window closes block scheme by exterior orientation automatically (to build initial block layout).



Code	Name	X, m	Y, m	Z, m	Omega, °	Phi, °	Kappa, °	Time	Type (XYZ)	Type (angles)	Std. dev. X, m	Std. dev. Y, m	Std. dev. Z, m	Std. dev. Omega, °	Std. dev. Phi, °	Std. dev. Kappa, °
42	DMCI...	396...	607...	111...	0.018450000	-0.029090000	-1.698240000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
43	DMCI...	396...	607...	112...	-0.005820000	-0.023110000	-1.689320000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
44	DMCI...	396...	607...	112...	0.081230000	-0.009790000	-1.703810000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
45	DMCI...	397...	607...	113...	0.006230000	-0.022830000	-1.692000000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
46	DMCI...	397...	607...	113...	0.017600000	-0.025790000	-1.682100000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
47	DMCI...	397...	607...	113...	0.002690000	-0.025270000	-1.677720000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
48	DMCI...	397...	607...	113...	-0.068280000	-0.032180000	-1.677010000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
49	DMCI...	397...	607...	113...	-0.038240000	-0.024750000	-1.681350000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
50	DMCI...	398...	607...	114...	-1.667160000	0.080030000	-1.676940000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
51	DMCI...	398...	607...	114...	-0.005600000	-0.035940000	-1.689240000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
52	DMCI...	398...	607...	114...	-0.004250000	-0.027450000	-1.668890000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
53	DMCI...	398...	607...	113...	-0.076600000	-0.017860000	-1.669080000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
54	DMCI...	398...	607...	113...	-0.007960000	-0.029260000	-1.663130000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
75	DMCI...	398...	607...	114...	0.002460000	-0.021420000	-1.667740000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
76	DMCI...	398...	607...	114...	0.039920000	-0.041260000	-1.673280000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
77	DMCI...	398...	607...	114...	0.107160000	-0.047140000	-1.669490000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
78	DMCI...	398...	607...	113...	-0.089670000	-0.008760000	-1.690610000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
79	DMCI...	398...	607...	113...	-0.018620000	-0.012540000	-1.679400000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
80	DMCI...	397...	607...	113...	0.085680000	-0.040960000	-1.677990000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
81	DMCI...	397...	607...	114...	0.018630000	-0.021840000	-1.679830000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
82	DMCI...	397...	607...	115...	0.031920000	-0.040370000	-1.674700000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
83	DMCI...	397...	607...	115...	-0.070840000	-0.026660000	-1.686980000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
84	DMCI...	397...	607...	114...	0.069190000	-0.033510000	-1.706110000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
85	DMCI...	396...	607...	114...	0.044820000	-0.014050000	-1.671880000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
86	DMCI...	396...	607...	114...	0.057070000	-0.008920000	-1.717750000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000
87	DMCI...	396...	607...	113...	-0.116970000	0.028080000	-1.654940000		Control	Check	0.2000000	0.2000000	0.1000000	0.005000000	0.005000000	0.005000000

Fig. C.14. List of exterior orientation parameters

The program provides possibility to create a block scheme by UAS images if angular exterior orientation parameters of block images are specified with low accuracy or are unavailable. For this operation it is necessary to use the button of the **Exterior orientation parameters** window toolbar:

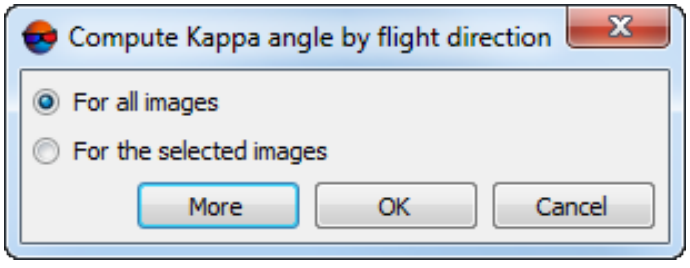


Fig. C.15. The Compute Kappa angle by flight direction window

Specify how to **compute Kappa angle by flight direction** – **for all images** of **for the selected images** only. For each highlighted image the program calculates the direction from previous to next center in a strip (for utmost images of a strip – from adjacent image to the current one) and kappa angle is setup from this direction.

If the survey time is available the correct images order is checked, otherwise images in strips should be located in chronological order only.

In order to eliminate blunders in block scheme building by exterior orientation parameters, perform the following actions:

1. Refine block layout parameters, by performing the following actions:

    1. Choose **Block › Block layout**. The **Block layout** window opens:

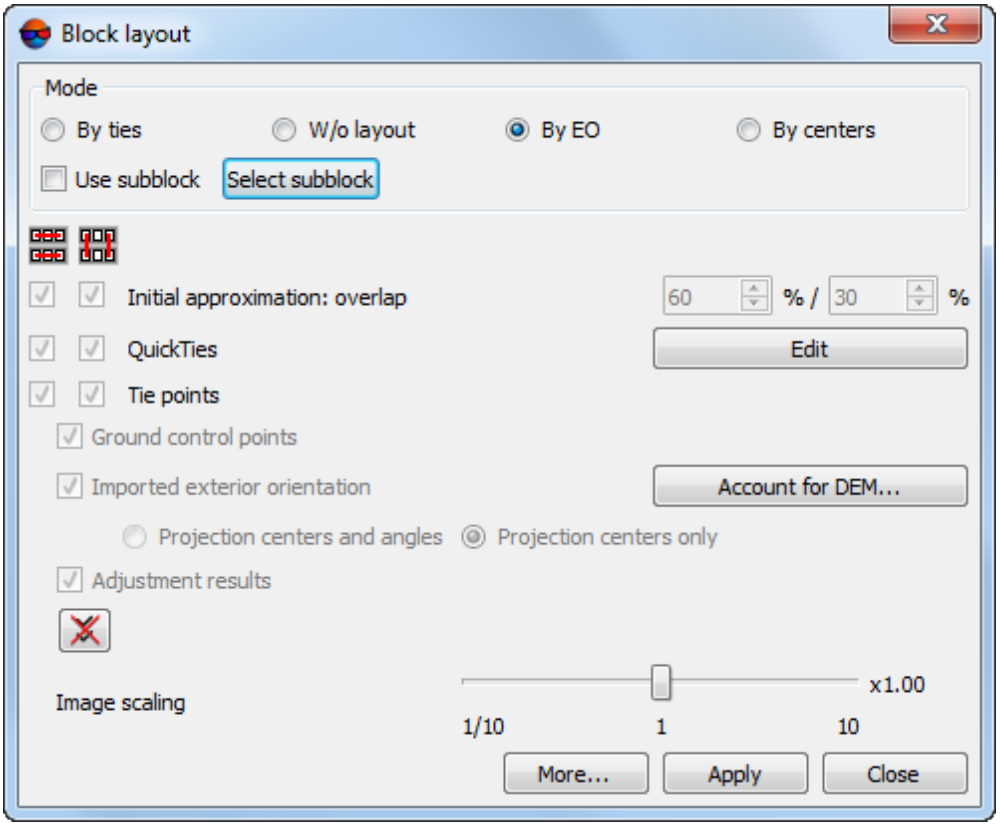


Fig. C.16. The Block layout window

    2. Choose the **By EO** mode and click the **Apply** button.

See a description of block layout creation in the “Aerial triangulation” User Manual.

2. Import exterior orientation parameters from a file of text format once more and refine import parameters.

3. To split images into strips by imported exterior orientation parameters (if any), perform the following:

1. Choose **Block › Split into strips › By exterior orientation data**. The **Split into strips by exterior orientation** window opens.

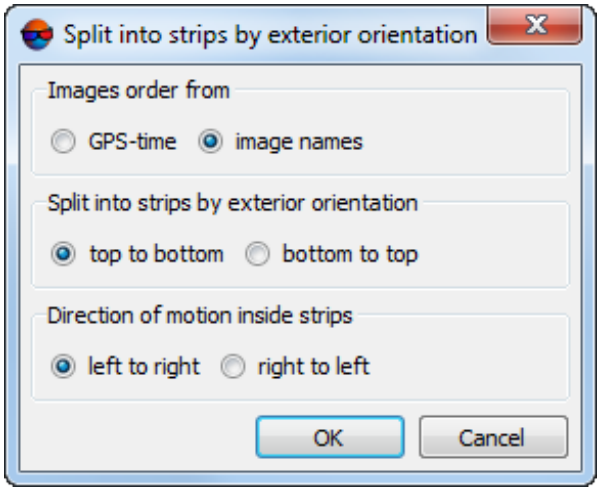


Fig. C.17. Split images into strips by exterior orientation

2. Specify the following parameters of splitting images into strips:

   - **Images order from** by GPS-time or image names;

     If the GPS-time is unavailable for part of images, they add to the separate strip automatically;

   - **Direction of motion between strips** – up to down or bottom up;
   - **Direction of motion inside strips** – left to right or right to left.

3. Click OK. Block scheme rebuilds by set parameters.

Block scheme, created using exterior orientation is an initial block layout and requires further refining by tie/GC points (see Section C.4.1).

C.4. Relative orientation

This chapter describes the process of automatic creation of tie points using the correlator. For a detailed description of the relative orientation of images (including a description of creating and editing tie points manually), see the “Relative orientation” section of the “Aerial triangulation” User Manual.

One of the stages of building a triangulation network is images relative orientation, that requires measured coordinates of tie points.

Tie points – the same points on the terrain on adjacent block images needed to build single models from stereopairs and then to merge them into triangulation networks.

After measurement of tie point coordinates it is necessary to perform relative orientation of block images, to input and measure GCP coordinates, to perform network adjustment and to obtain exterior orientation parameters of a block.

Measuring of tie point coordinates – is stereoscopic measurement, i.e. measuring of coordinates of the same point on both stereopair images at once (or on multiple overlapped adjacent images) in stereomode.

C.4.1. Automatic measurement of tie points coordinates (general information)

The system provides possibility to measure tie points coordinates and perform relative orientation in automatic mode. When working in automatic mode correlation algorithm is used for search and measurement of tie points coordinates.

C.4.1.1. Required data

To perform automatic measurement of tie point coordinates the following data should be available:

- interior orientation parameters of images, which will be considered in automatic measurement of points coordinates;

  It is highly recommended to perform interior orientation prior to automatic search of tie points, since otherwise most of the filtering parameters are ignored, that considerably reduces relative orientation quality.

- block layout (see the “Aerial triangulation” User Manual).

  Sufficient condition for launch the operation of automatic measurement of tie points coordinates is availability of initial (“rough”) block layout, considering, for example, exterior orientation data (see above) and/or quick ties data, or size of images overlap.

  However, the more data is used for the block layout creation, the more accurate is the block scheme and, hence, the better the results of measurement of tie points coordinates in automatic mode.

C.4.1.2. Modes of automatic measurement of tie points coordinates

The **Automatic tie points measurement** (**Orientation › Automatic tie points measurement**) window have different configurations, provided to process various data types:

When working with airborne data the window allows to select manually optimal combination of basic and additional parameters of automatic ties search, measurement and filtering, and also to save sets of this parameters as files with `*.x-ini` extension for the further usage (see the “Aerial triangulation” User Manual).

When working with UAS and oblique data the window allows to load parameters sets for automatic ties search, measurement and filtering (so called “presets”).

Preset – parameters set of automatic tie points measurement, preliminary created by user or setup in advance.

C.4.1.3. General order of automatic tie points measurement

To measure tie points using automatic mode (на примере airborne data) perform the following:

1. Create a block layout considering available data.

   Parameters of inter-strip stereopairs forming have a significant impact on automatic tie points measurement and on further adjustment.

   You can configure parameters of inter-strip stereopairs forming in the **Parameters** window (**Service › Settings**) on the **Block scheme** tab, **Forming interstrip stereopairs** section.

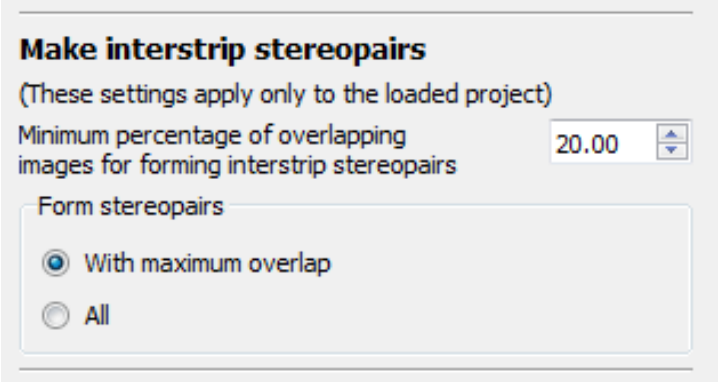


Fig. C.18. The “Forming interstrip stereopairs” section of the “Block scheme” tab of the “Settings” window

   - When specifying the **Minimal images overlap for forming interstrip stereopairs** parameter, the user should correctly estimate the interstrip overlap value (in case of “bad” overlap high values of this parameter will lead to the fact that interstrip ties will not be measured);

   - The **Form stereopairs** parameters allows to choose, which stereopairs with specified overlap will be formed – **all** or **with maximum overlap**. If the **all** option is selected, it allows to considerably increase the number of measured interstrip points, with significantly increased time costs.

2. Setup parameters of tie points automatic measurement, transfer, filtering and start the operation of automatic measurement of tie points coordinates;

3. Perform accuracy estimation of relative orientation and detecting of tie points measurement errors;

4. Manually eliminate errors of tie points measurement in the **Points measurement** module or choose other filtering parameters (for example, specify more “strict” tolerances for measurement errors) and start the operation of automatic measurement of tie points coordinates once more;

5. Update block layout considering tie points. Change settings of automatic tie point measurement parameters if necessary and start once more operation of further collecting of tie points measurements. Repeat steps 3-4 until obtain satisfactory results.

C.4.2. Automatic tie points measurement on UAS data

To measure tie points coordinates in automatic mode perform the following actions:

1. Choose **Orientation › Automatic tie point measurement › UAS**. The **Automatic tie points measurement** window opens.

   For automatic measurements all images are selected by default. For tie points search just on selected images clear the checkbox **Select all images for automatic tie point measurement** on the **Orientation** tab of the **Parameters** window.

2. Use default settings of measurement parameters (*presets*) or set manually the main parameters for tie points search, measure and filter.

   The recommended default preset is **Optimal**.

3. [optional] Specify the adjustment parameters, if it is necessary to perform the adjustment on the current project step, otherwise – clear the **Make block adjustment** checkbox.

4. [optional] In order to use distributed processing during measurement of tie points coordinates, perform the following actions:

   1. Change settings and run the distributed processing server/client (see the “Distributed processing” chapter in the “General information about system” User Manual).

   2. Click the **Distributed processing...** button. The **Distributed processing settings** window opens.

Specify the **Maximum number of parallel tasks** value is displayed in the window. This parameter shows how many processing tasks is possible to divide the current project to.

In case the workstation processor supports the *hyperthreading technology*, it is recommended to reduce the **maximum number of parallel tasks** by half in order to ensure system's stability.

Hyper-threading (hyper-threading technology) is simultaneous multithreading implementation used to improve parallelization of computations. With HTT, one physical core appears as two processors to the operating system, allowing concurrent scheduling of two processes per core.

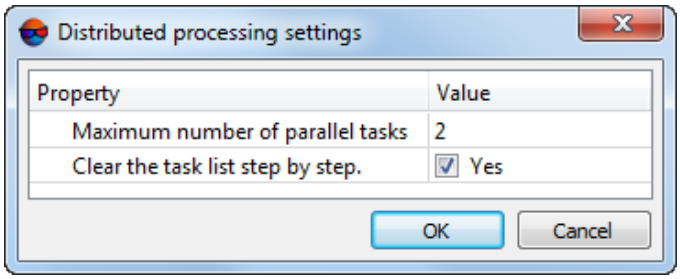


Fig. C.19. Parameters of distributed measurement of tie points in automatic mode

3. The system allows to **clear the task list step by step** in the distributed processing monitor (tasks are removed from the list as they are successfully completed).

   Clearing the task list makes it possible to limit the increasing size of the distributed processing database, what can significantly affect system performance.

   It is strongly suggested to clear the task list when processing large amounts of data on workstations with limited disk space.

   Full display of the task list (for further analysis of their logs) may be feasible when restarting calculations, if the previous data processing session failed.

4. Click OK to go back to previous window.

   The Progress bar remains open during the entire operation running, allowing the user to directly monitor the progress of data processing from the *PHOTOMOD* interface without opening the **Monitor for distributed processing** (see "Progress bar for distributed processing" in the "General information" User Manual).

5. Click the **OK** button in the **Automatic tie points measurement** window to start operation of automatic measurement of points coordinates.

6. The *processing report* (see the appropriate chapter in "Aerial triangulation" User Manual) opens automatically after the operation is completed, if after the completion of automatic relative orientation the adjustment was carried out (see the **Make block adjustment** checkbox below).

User also can open the *relative orientation report* to view results of relative orientation operation and to fix measurement errors (see the appropriate chapter in "Aerial triangulation" User Manual).

C.4.2.1. Main parameters of automatic tie points measurement

The **Automatic tie point measurement** window allows to use parameters presets of tie points measurement, or to manually choose the optimal combination of basic and additional parameters for automatic tie points search, measurement and filtering.

Choose **Orientation › Automatic tie point measurement › UAS**. The **Automatic tie points measurement** window opens.

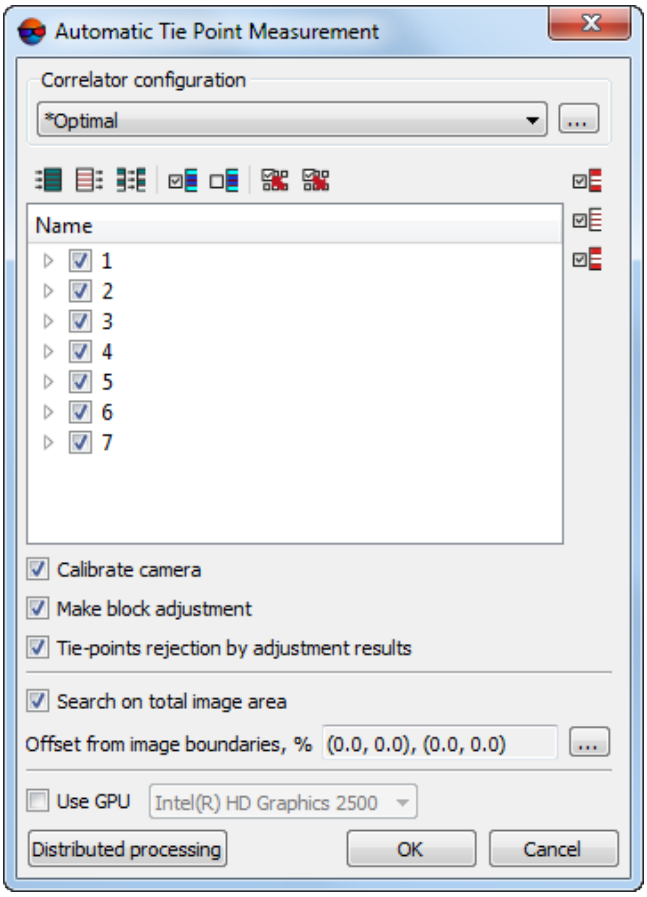


Fig. C.20. The Automatic tie points measurement window

The **Automatic tie points measurement** window includes the following interface elements:

- the images selection toolbar used to open auxiliary windows, to save and load sets of parameters;
- a list of project images allows to select images, scheduled to perform automatic measuring of tie points coordinates;
- the **Correlator configuration** section is used to load, create and edit sets of parameters of automatic tie points measurement;
- the **OK** button used to launch an operation of tie points measurement and to perform relative orientation in automatic mode;
- the **Distributed processing** button is used for automatic tie points measurement in distributed processing mode.

The system provides the following buttons in the toolbar to work with auxiliary windows:

- – allows to select stereopairs where there is no tie points coordinates measurements, and their adjacent images;
- – allows to select triplets without tie points;
- – allows to select *interstrip* stereopairs where there is no tie points coordinates measurements;

When searching for stereopairs and triplets, excluded ( ) points are also not taken into account. The `Excluded` status (in this case) is equivalent to no measurements. The user can change the status of a point using the appropriate tools of the **Triangulation points** window.

To configure automatic tie points measurement and (optionally) for further adjustment procedure the following parameters are used:

- [optional] clear the **Calibrate camera** checkbox if camera calibration has already done and is not needed;

It is recommended to clear the **Calibrate camera** checkbox only if there is reason to trust the camera calibration. It is necessary to consider, that if the calibration was not carried out in actual operating conditions, then significant discrepancies in the obtained calibration parameters may occur.

If *polyplets* (sets of several overlapped images located on different strips) are not found, one of camera calibration stages (calibration by polyplets) will be skipped (an appropriate warning message will appear). Other calibration stages will be carried out.

In certain cases, since some coefficients can be dependent, it is impossible to calculate corrections to them (during camera calibration, before performing adjustment). In such cases, some of these coefficients can be automatically excluded from the calibration process (but not from the adjustment process) i.e. during data processing, the initial coefficient value will be used, without introducing corrections. An appropriate info message is to be displayed in

the progress bar (see the “Loading progress window” section in the “General information” User Manual).

- [optional] clear the **Make block adjustment** checkbox if the adjustment operation is not necessary at this stage of the project;
- [optional] clear the **Search on total image area** checkbox, to exclude from tie point search those areas of a block where there are no image overlap according to rough block layout results. This will allow to increase the system speed and improve the correlator accuracy.

This checkbox is recommended only if primary block layout was performed correctly, after visual estimation of image positions.

In case of aerial survey data processing, primary block layout is as usual quite accurate for imagery provided with correct elements of exterior orientation and taken with off-nadir angle no more than 15 degrees.

  - In order not to use boundary regions of imagery while search, set the **Offset from image boundaries, %** by clicking the ... button. The **Image margin** window opens. Set image margins in percent which are not to be involved in points search;

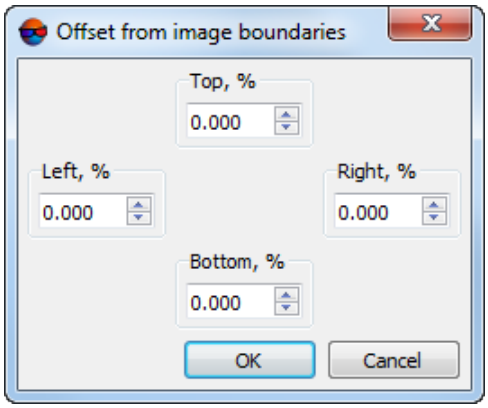


Fig. C.21. The Image margin window

- To increase the system performance through graphic processing unit resources, set the **Use GPU** checkbox and select the desired device from the drop-down list;

The intensity of GPU usage can be monitored using various free software. *Windows 10* users can monitor GPU activity through the **Windows Task Manager** (**Performance** Tab).

It is recommended to use modern graphics adapters with at least 4-6 GB of memory, especially in the case of parallel execution of several tasks in distributed processing mode.

It should be considered that the performance of a video adapter as an extra device for computations is not directly dependent on its memory amount but is influenced by the algorithms used and features of the system architecture.If there is a choice, first of all, *NVidia* video adapters are preferable.

During distributed processing, video adapters are to be selected in the **Monitor for Distributed Processing** window, individually for each computer used as a distributed processing client (see "Computers" in the "General information" User Manual. However, video adapters' resources will be used for distributed computations only if the **Use GPU** checkbox is set in the current window.

If the computer used to configure the operation options (*server*) is also used as one of the *clients*, the selection of the video adapter used by it is also carried out through the **Monitor for Distributed Processing**, regardless of which device was selected in the drop-down list in the current window (this choice is taken into account only in the case of standard data processing mode).

The **Correlator configuration** section is used to load, create and edit *presets* – set of parameters of automatic tie points measurement;

- the download button in the right part of the **Correlator configuration** section allows to open the **Correlator precision** window, which contains a list of available (both predefined and user-created) presets and the following buttons:

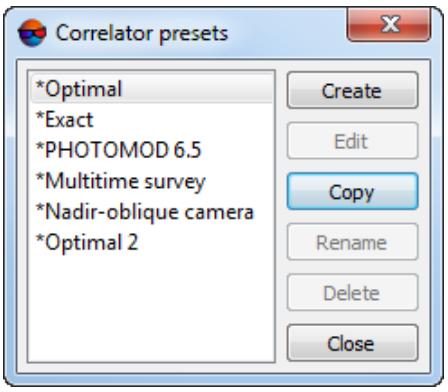


Fig. C.22. The Correlator presets window

  - **Create** – allows to create a custom preset – set of parameters of automatic tie points measurement. Click the **Create** button to open the **Enter entry** window, where user should enter a name for a preset to be created (default *user preset*).

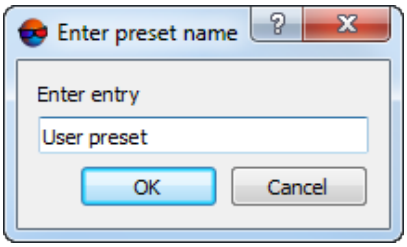


Fig. C.23. The Enter entry window

It is not allowed to create presets with identical names. When you try to create presets with identical names, the system opens a window with a corresponding error message.

Clicking the OK button opens the **Edit parameters** window, where you should manually specify the following parameters:

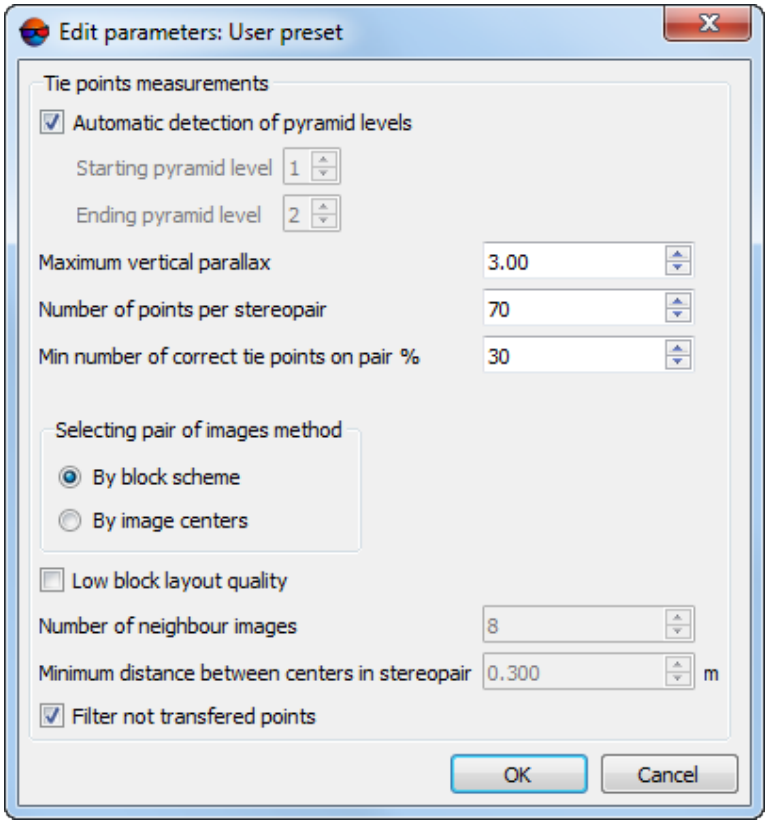


Fig. C.24. The Edit parameters window

- **Starting pyramid level** – it is recommended to increase the value of starting pyramid level when system performance is insufficient, and when there is no high requirements to processing results accuracy. Recommended value of starting pyramid level is 1. The value of the starting pyramid level that equals to zero provides maximum accuracy of the results, but leads to increasing of data processing time;

- **Ending pyramid level** – it is recommended to increase the value of ending pyramid level when using data with different resolution (including cases of significant difference in spatial resolution within *one* image);

  For example, when using data which resolution varies 2 times, it is recommended to increase the default value of the ending level of the pyramid on the one.

- **Maximum vertical parallax** (in pixels or mm) – threshold value for rejection executed after points measuring/transfer. On each step of rejection process, the program searches for the point with maximal Y-parallax on the stereopair and rejects it. After that, it makes relative orientation once more and the iteration is

executed until maximal Y-parallax exceeds the parameter specified here, or until 6 points or less remain on stereopair. Recommended value is a pixel size;

- **Points per stereopair** – allows to specify a number of points to be measured on in-strip and inter-strip stereopairs;

The resulting number of measured points may be both greater than a predetermined amount – due to points transfer from adjacent stereopairs, and less due to the subsequent rejection of points. Default value is 30 points on each in-strip images pair, and 10 points – in inter-strip overlap. The minimum number of points to control the vertical parallax residual is 6.

- **Minimal correct matching points in pair**, in percent allows to exclude from the processing “bad” stereopairs that do not match the given values. The increase of this parameter can improve the system performance by excluding a portion of data from the processing;

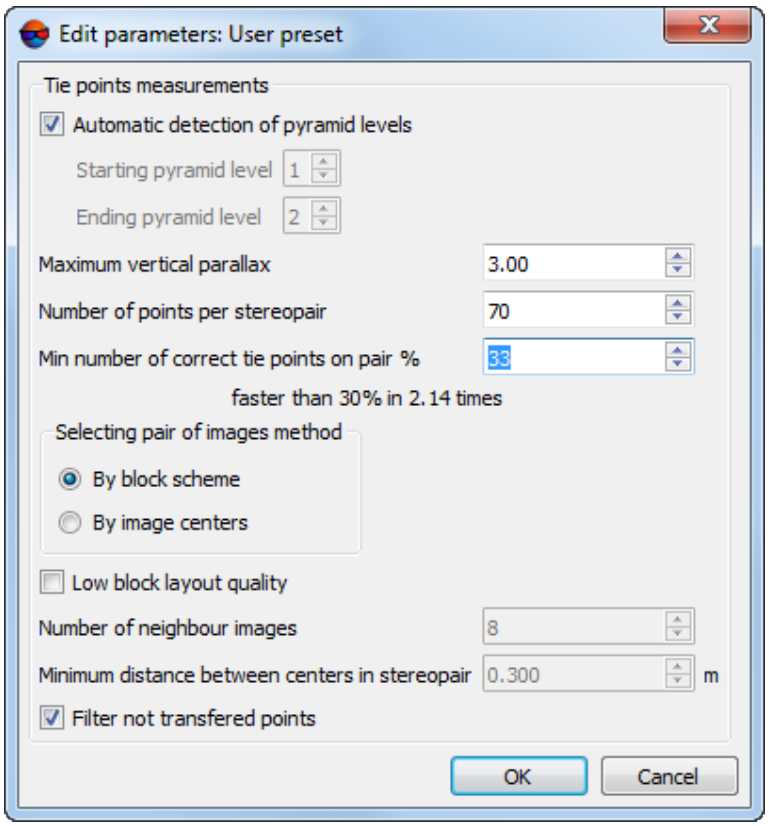


Fig. C.25. The Edit parameters window

Default value of this parameter is 30%.

When the default value increase, an approximate estimation of the system speed change is displayed – **faster than 30% in ... times**.

When the default value decrease, an approximate estimation of the system speed change is displayed – **slower than 30% in ... times**.

- Set the **Filter non-transferred points** checkbox on to reject points located on inter-strip stereopairs. A point will be filtered, if the system recognizes it on a single image of some strip.

Set the image pair selection method:

- [optional] **by block scheme**;

  The **Low block layout quality** checkbox allows to divide the process of stereopair search **by block scheme** into several iterations, thereby increasing the accuracy of final results.

- [optional] **by image centers** – set the **number of neighbour images** and **minimum distance between centers in stereopair**.

  In case of the processing of quite high-quality data with satisfactory overlap, limiting the number of adjacent images leads to a significant increase of data processing speed, without affecting the accuracy of the final results.

  Limiting the minimum distance between two centers in a stereopair allows to exclude from processing stereopairs with projection centers that are too close to each other.

When you click OK, the system creates a new user preset – a set of parameters used for automatic tie points measurement.

- the **Edit** button allows to open the **Edit parameters** window to change settings of the selected preset.
- the **Copy** button allows to copy a preset.
- the **Rename** button allows to rename a preset.
- the **Delete** button allows to delete a preset.
- the **Close** button allows to close the **Correlator precision** window.

- The arrow to the left of the button which opens the **Correlator precision** window allows to open a drop-down list, used to select one of the following presets.

  The system provides a set of predefined presets, allowing to vary the settings of automatic tie point measurements. Predefined presets are marked by the asterisk symbol *.

The system does not allow the user to edit the predefined presets. A copy of the predefined preset is available for editing, as well as a custom preset.

C.5. Ground control points

This chapter addresses importing a GCP catalogue from a text file and manually measuring GCPs in images. For detailed information on GCPs (including manually entering geodetic coordinates of points and automatically measuring GCPs in images), see the "Aerial triangulation" User Manual.

GC point is a terrain point with known geodetic coordinates, which is a source point for photogrammetric measurements.

Measurement of GC point coordinates means adding of known terrain point coordinates to GC points list and measurement of its pixel coordinates on image. In order to use GC points during block adjustment perform the following actions:

1. Import coordinates of GCP to coordinates list.
2. Measure coordinates of GC points on project images.

To perform correct block adjustment it is recommended to measure coordinates of all input GC points on block images, since all points – both measured and unmeasured – take part in statistics during block adjustment. Otherwise, it is possible to exclude unmeasured points on steps of coordinates measurement and block adjustment.

Adding and measurement of GC points coordinates on block images in the system allow to calculate exterior orientation parameters of images block, when metadata with exterior orientation parameters, obtained from a provider, is not available.

If the GC points coordinates data is unavailable at the time of data collection for aerial triangulation procedure, the system provides possibility to adjust a block in a *free model* (without GC points), that allows to proceed to project processing step directly (see the "Block adjustment" User Manual).

When deleting images from a project it is highly recommended to synchronize data in a project to remove redundant results of orientation. To do this choose **Project › Synchronize**.

C.5.1. Import of GCP catalogue

The system provides import of GC points coordinates list from file with `*.csv` or `*.txt` extension.

Refer to the "Import of triangulation points from X-points" chapter of the "Aerial triangulation" User Manual for the description of the GC points import from file with `*.x-points` extension.

To perform import of GC points coordinates list from file perform the following actions:

1. Click the **Import catalog** button on the **Aerial Triangulation** tab of the main toolbar.

2. Select a file containing GCP catalogue and click OK. The **Import GCP catalogue** window opens.

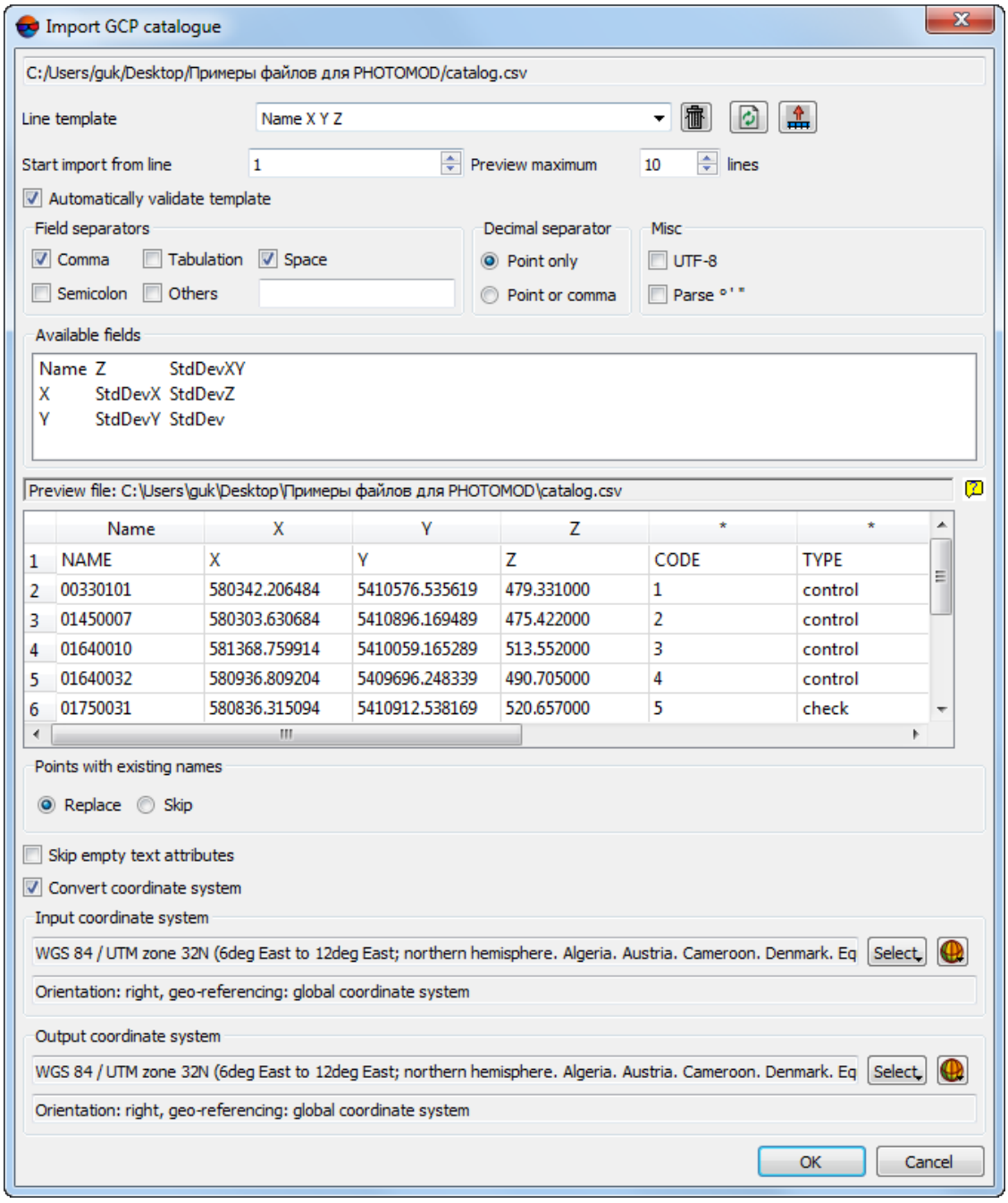


Fig. C.26. Import of GCP catalogue

3. Make sure that the **Line template** was configured correctly.

The **Line template** field displays a list of main and additional fields, which must be located in each line of the import file:

- `Name` – object's name;

If imported file does not contain a column with GCP names (numbers), then after import operation the names like *Point0001* are assigned to the points automatically.

- `X`, `Y`, `Z` – values of GC point coordinates by X, Y, Z axes;
- [optional] `STDDEVX`, `STDDEVY`, `STDDEVZ` – accuracy of coordinates measurements by X, Y, Z axes (RMS;
- [optional] * – marks columns with data which is not imported.

The **Preview file** table contains data of imported file. Fields type according to the template, located in the **Line template** field, are automatically assigned to the table columns.

All objects are saved using the same template. Each line of a file contains the same number of fields, that equals to number of fields in template. Lines which does not correspond to the template, are skipped. To all vertexes two (for 2D objects import) or three coordinates are specified.

In order to begin the active template setup, perform the following:

- [optional] In order to display necessary number of lines in the **Preview file** table, set the **Preview maximum** parameter. 10 lines are displayed by default.
- [optional] In order to define a line from which to start data import, input the line number to the **Start import from line** field.

In many cases, the imported file may have a first row containing headers describing the contents of the columns in the file. Accordingly, this row should not be included in the imported data, but its display in the **Preview file** table may be useful when setting up the row template. It is recommended to set this parameter at the very end.

- In the **Field separators** section set single or multiple checkboxes to specify field separators in the file: **comma**, **space**, **tab**, **semicolon** or **other**. The comma and space are set by default.
- In the **Decimal separator** section set the following parameters:
  - **Point only** – to use only a point as a decimal separator in coordinates;
  - **Point or comma** – to use both a point and a comma as a decimal separator in coordinates.

If the **Field separator** is selected as a comma, it is not recommended to specify the **Decimal separator** as a comma, since objects with incorrect coordinates will be created after import operation.

- [optional] In the **Misc** section set the following checkboxes:
  - **UTF-8** – is used to recognize text in Unicode coding;
  - **Parse degrees minutes seconds** – is used to recognize records from the list of projection centers or GCP.

When using this parameter it is highly recommended to check recognizing correctness after import operation.

In order to finish the active template setup, perform one of the following actions:

- drag a field name from the **Available fields** list to the **Preview file** table column header. After that the template in the **Line template** field is changed. To cancel a field selection, double click the **Preview file** table column;
- change a template manually in the **Line template** field. After that column types in the **Preview file** table will be changed automatically.

The  button is used to return to default template `Name, X, Y, Z`.

The  button is used to compare the **Line template** field with data shown in the **Preview file** table.

The  button is used to replace specified fields names with fields names from the first line of the **Preview file** table. It is possible to specify any names for vectors import.

For import of laser scanning data the field names from the list of available names are specified.

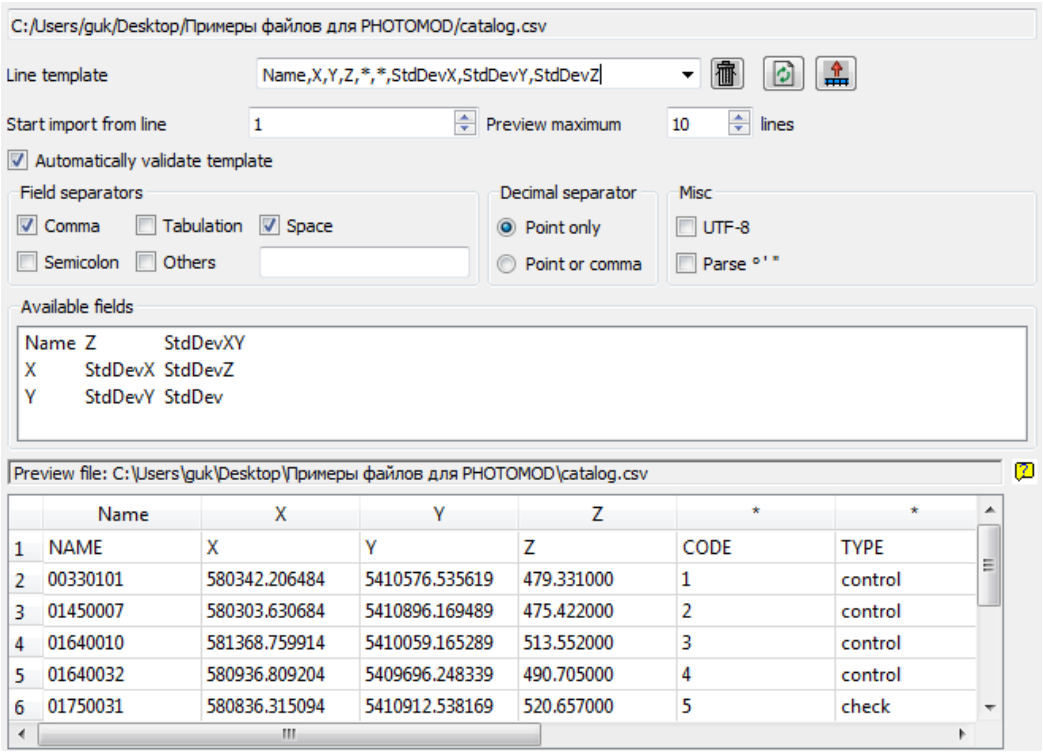


Fig. C.27. Setup of a template sample

For automatic selection of current template the **Automatically validate template** checkbox is set by default. In order to setup a template for a file that contains lines with different number of columns, clear the **Automatically validate template** checkbox and setup the template manually in the **Line template** field.

4. In the **Points with existing names** section specify actions to perform when names of imported GC points coincide with points names already existing on the **GCP list** tab:

   - **Replace** – allows to replace a point with the same name in the list, i.e. the point data will be updated;
   - **Skip** – allows to cancel import of GCP with the same name.

5. [optional] To prevent creating text attributes without value during import, set the **Do not create empty text attributes** checkbox on.

   When the **Do not create empty text attributes** checkbox is on, in some cases a data loss may occur.

6. [optional] To convert coordinate system set the **Convert coordinate system** checkbox and specify source and output coordinate systems.

7. Click OK. The **Import ground control points** window opens, which contains the GCP table to be added. Points which names are the same as names of existing GCP in the GCP list of the **Triangulation points** window are shown by red color, new GCP – by black color.

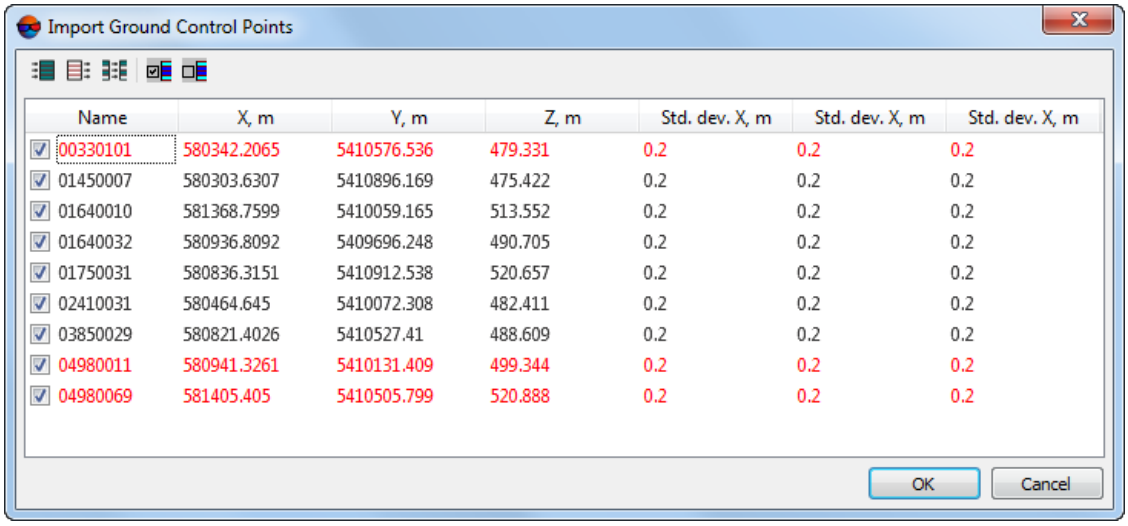


Fig. C.28. Selecting GCP to be added to the list

8. Select points to be imported using checkboxes and/or standard selection tools. Click OK. After that coordinates of GC points with set checkboxes will be loaded in the end of the GCP list.

If in the **Points with existing names** section the **Skip** parameter is selected, then only new GCP are selected for adding by default, if the **Replace** parameter is selected, all GCP checkboxes are selected to be added.

If RMS of X, Y, Z coordinates was not imported or is absent in GCP file, then default value of RMS for all coordinates is 0.2 meters.

C.5.2. Measurement of GCP coordinates

Refer to the “Measurement of GCP coordinates” chapter of the “Aerial triangulation” User Manual for the detailed description.

After completing of GCP coordinates input it is possible to measure these GCP on block images in the *Points measurement* module.

In order to measure GC point coordinates perform the following actions:

1. Open the **Triangulation points** window on the **GCP list** tab.

2. Select GC point in the table to measure its coordinates on images and click the button or double click the point name in the table. The *Points measurement* module is opened.

   If at least three GCP were already measured on block images or there was executed preliminary orientation in geodetic coordinate system (by projection centers, for instance), the system performs automatic search for corresponding terrain point on block images (according to XY coordinates) and places marker on this image point.

   When the system performs a search for terrain point in mode of images automatic selection (the button of the **Triangulation points** window is clicked) it automatically opens all images that contain the terrain point with measurements or without them, otherwise – only images that contain measurements of selected point.

3. Measure coordinates of GC point on a single or multiple images (the point lies in the overlap area) using one of the following ways:

   - *GCP coordinates measurement without correlator* is used if a point is located on terrain and is clearly recognized on all images.

     Show by marker the terrain point on images. Click the button of the upper toolbar of the module.

   - *GCP coordinates measurement and transfer with correlator* is used if a point is not clearly recognized on all images.

     In this case measure the point on a single image using the button of the upper toolbar of the image window or use the **Ins** hotkey. Define the image as the left using the L button in the image window. Then transfer the point from the left image to other images using the button to transfer the point on the image in the active window or use the button to transfer the point on all images opened in the module.

     Recommended correlation coefficient during GCP transfer is 0.90–0.95.

   - *measurement of coordinates in stereomode* is used when the point is located on high-rised object and/or is not displayed on all images.

4. After measurement of GCP coordinates an information about the measurement appears in the **Tie points** list.

To move a GCP on images (to edit measurement of selected point) perform the following actions:

1. Select measured GC point in the table of the **Triangulation points** module and click the button or double click the point name in the table to display the measurements of selected point on block images.

2. Show by marker new position for selected GC point.

3. Click the of the module upper toolbar. After that the GC point on images is moved to marker's position, and measurements data is refreshed in the **Triangulation points** table.

To delete the ground control point *measurements from the image*, select the point in the **Triangulation points** table and click the button of the image window in the **Points measurement** module.

To delete the ground control point *measurements* from *all images*, perform one of the following actions:

- select a point in the **Triangulation points** table and click the button on the *upper toolbar* on the **Points measurement** module;

- select a point in the **Triangulation points** table and click the button on the toolbar of the **Triangulation points** window. The **Delete points** window opens. Click the **Delete point measurements only** button.

The operation does not lead to deletion of GC point coordinates from the **Triangulation points** table.

To delete a ground control point completely (both the point measurements on the images and the point with the coordinates from the table), select the point in the table and click the button of the toolbar of the **Triangulation points** window. The **Delete points** window opens. Click the **Delete points from catalogue** button.

C.6. Adjustment

Refer to the "Block adjustment" User Manual for the detailed description about block adjustment procedure.

C.6.1. Preliminary block adjustment

In order to adjust images block perform the following actions:

1. Click the button of **Block editor** window's toolbar to refresh initial block scheme (to refine block layout) using measured tie and GC points:

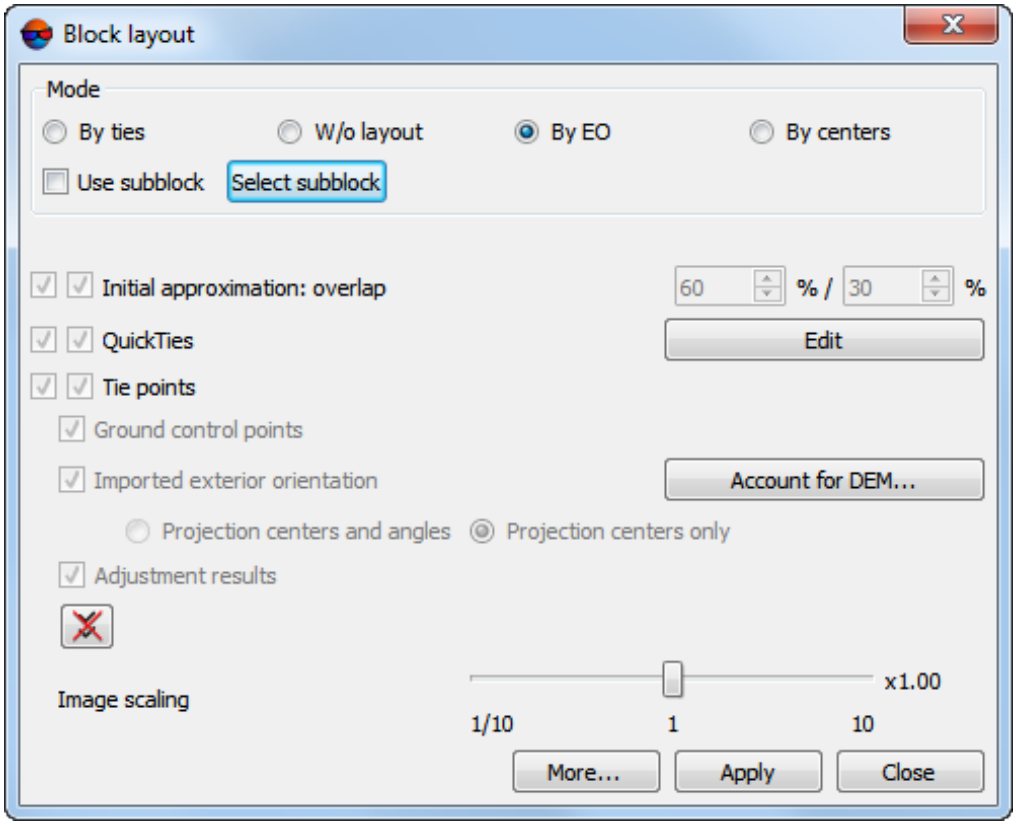


Fig. C.29. The Block layout window

See a description of block layout creation in the "Aerial triangulation" User Manual.

2. Choose **Orientation › Block adjustment** or click the **Block adjustment** button on the **Aerial Triangulation** tab of the main toolbar. The **adjustment** toolbar opens.

See detailed description of interior orientation step in the "Aerial triangulation" User Manual.

3. Click the button **Settings** on the **Adjustment** toolbar. The **Settings** window opens.

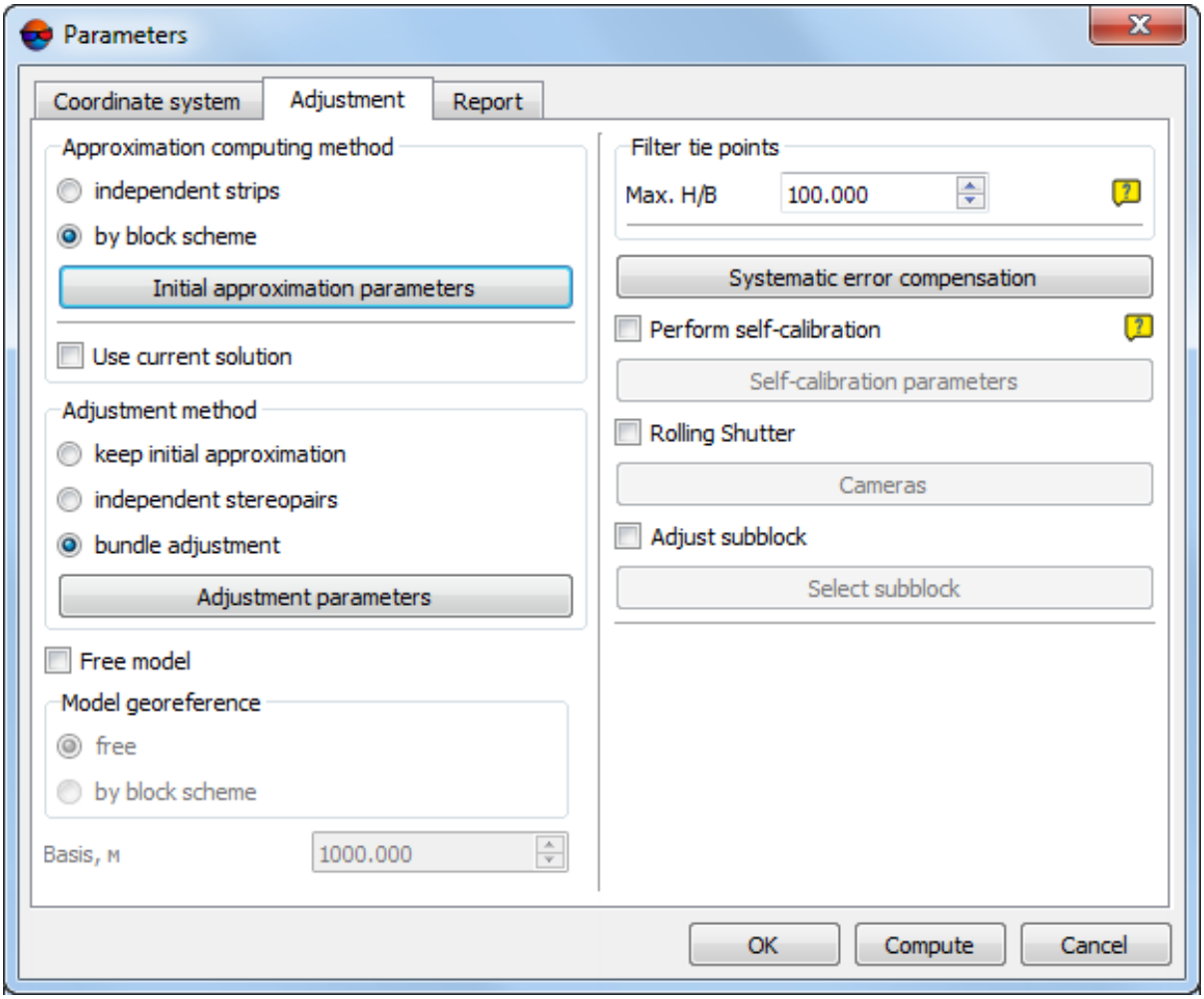


Fig. C.30. Adjustment parameters for the first adjustment operation

4. On the **Adjustment** tab define the following parameters *for the first* start of the adjustment procedure:

    1. Choose the **by block scheme** option in the **Approximation computing method** section.

    2. Click the **Initial approximation parameters** button. The **Initial approximation parameters** window opens:

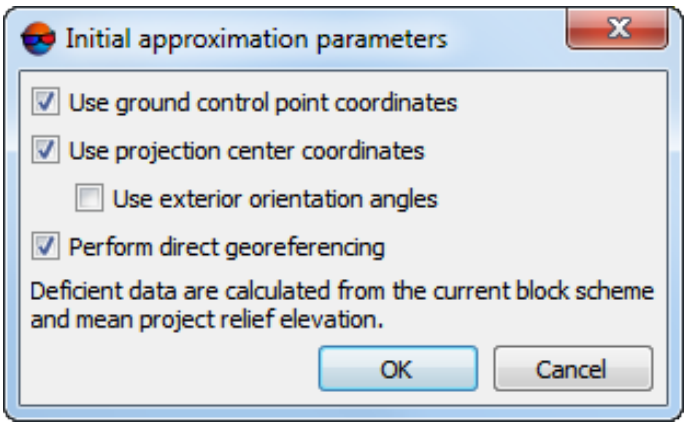


Fig. C.31. The Initial approximation parameters window

3. Set the **Use ground control points coordinates** and **Use projection centers coordinates** checkboxes.

4. Click OK.

5. Choose the **bundle adjustment** option in the **Adjustment method** section.

6. Click the **Adjustment parameters** button. The **Bundle adjustment parameters** window opens:

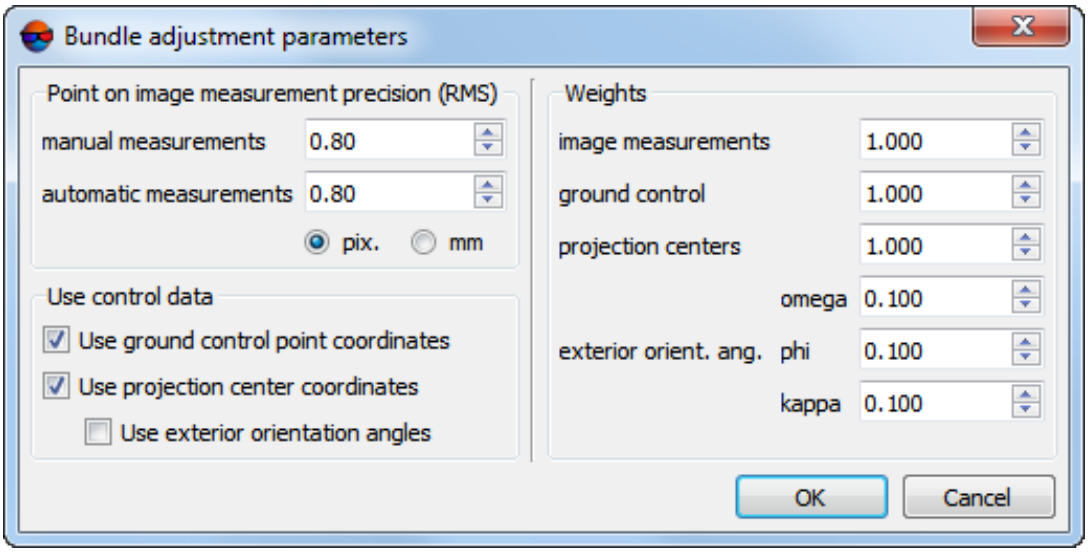


Fig. C.32. The Initial approximation parameters window

7. Set the **Use ground control points coordinates** and **Use projection centers coordinates** checkboxes.

8. [optional] To take the *rolling shutter* effect into account when adjusting a project, set the **Rolling shutter** checkbox and click the **Cameras** button. The **Cameras** window opens:

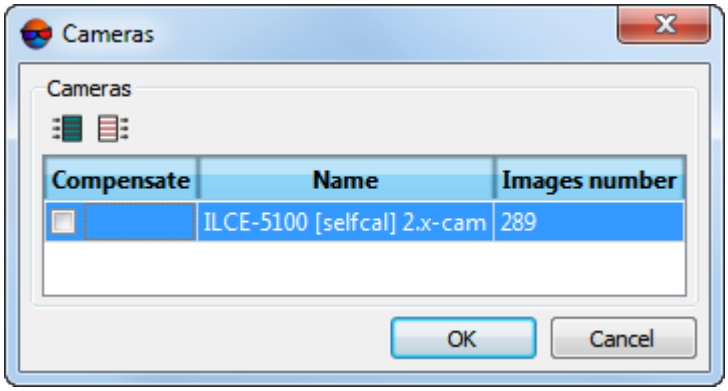


Fig. C.33. The Cameras window

Select cameras whose technical characteristics require *rolling shutter* compensation when adjusting a project that contains images taken with those cameras. Click Ok.

Click the button to select all cameras in the project. Click the button to deselect all cameras.

The effect of *rolling shutter* is the distortion of object shapes in an image when a picture is not captured at the same instant but the image is captured progressively. This effect appears at the short shutter speed of focal plane shutters or interline transfer CCD.

Survey data containing such distortions can be obtained using cameras installed aboard some UAS. It is recommended to use *rolling shutter* compensation algorithms if appropriate information about the properties of the equipment used is available.

9. Click OK to return for the **Parameters** window.

5. Click **Compute** to start the adjustment operation.

6. To analyse the adjustment results you can use adjustment report, visual control of residuals on block scheme and the **Residuals** window which contains brief residuals report.

7. Pick out adjustment parameters, analyse and eliminate residuals to obtain satisfying adjustment result.

After making any alterations to points measurements data it is necessary to re-start the adjustment procedure once again.

At subsequent start of the adjustment, the results of previous adjustment processes are used by default (the **Use current solution** checkbox on the **Adjustment** tab of the **Parameters** window).

The system allows to perform re-adjustment by changing any settings of bundle adjustment (in this case it is not recommended to clear the **Use current solution** checkbox, since that can lead to poor adjustment results).

8. It is recommended to use self-calibration of camera parameters during adjustment procedure, if camera without calibration was used.

C.6.1.1. Adjustment procedure

In order to perform adjustment, it is necessary to setup parameters of project adjustment and click the **Compute** button. The calculation procedure is started and the window opens where the adjustment operation status is displayed.

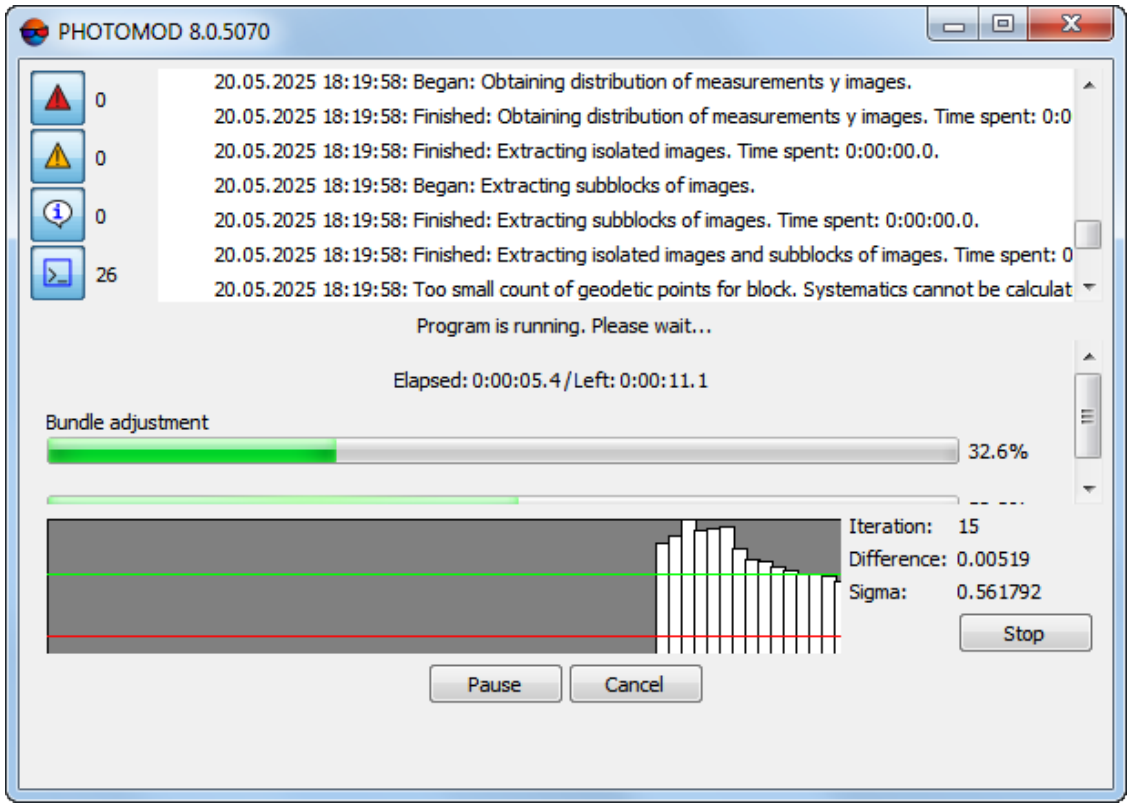


Fig. C.34. Window of adjustment procedure status

Adjustment operation convergence is represented by the diagram. The window also shows iteration number, adjustment method, and a value of refinement.

If the iterative process does not converge, interrupt the adjustment operation using the **Stop** button to display a result obtained on current iteration. In order to delete all the adjustment results click the **Cancel** button.

Non-convergent adjustment may be caused by relative orientation errors, errors in GCP coordinates or by incorrect adjustment parameters (see the “Stages of accuracy control” chapter in “Block adjustment” User Manual), or due to wrongly defined coordinate system (right-left).

Adjustment procedure interrupts only between iterations, at that the adjustment results are saved.

In order to save satisfactory adjustment results click the button of the **Block adjustment** toolbar. Pass to further project processing.

The button on the **Adjustment** toolbar is used to restore current block status. Also it allows to load adjustment results after project loading.

When adjustment brings unsatisfactory results and it is necessary to bring the block back to its initial status (as it was prior to adjustment launch), click the button on the **Adjustment** toolbar.

C.6.2. Using self-calibration

[optional] If a complete set of camera parameters is unavailable, use self-calibration of camera parameters during adjustment procedure.

Self-calibration of camera parameters – is automatic calculation of camera parameters (corrections to principal point coordinates and focal length, distortion coefficients) during adjustment operation.

In *PHOTOMOD UAS* system self-calibration of camera parameters option is set on by default.

To use self-calibration of camera parameters during adjustment operation perform the following actions:

1. Make sure that the **Perform self-calibration** checkbox on the **Adjustment** tab of the **Parameters** window is set on and click the **Self-calibration parameters** button. The **Camera parameters self-calibration** window opens.

   See detailed description of self-calibration operation in the “Block adjustment” User Manual.

2. Choose camera to be calibrated in the **Cameras** table.

3. Select the **Calibration type**: **Physical** or **Mixed**.

   For self-calibration of a camera having a focal plane shutter, it is recommended to use **Mixed** self-calibration.

4. In the **Coefficients** table define initial values of camera parameters in the **Initial value** column and choose parameters for optimization in the **Optimize** column.

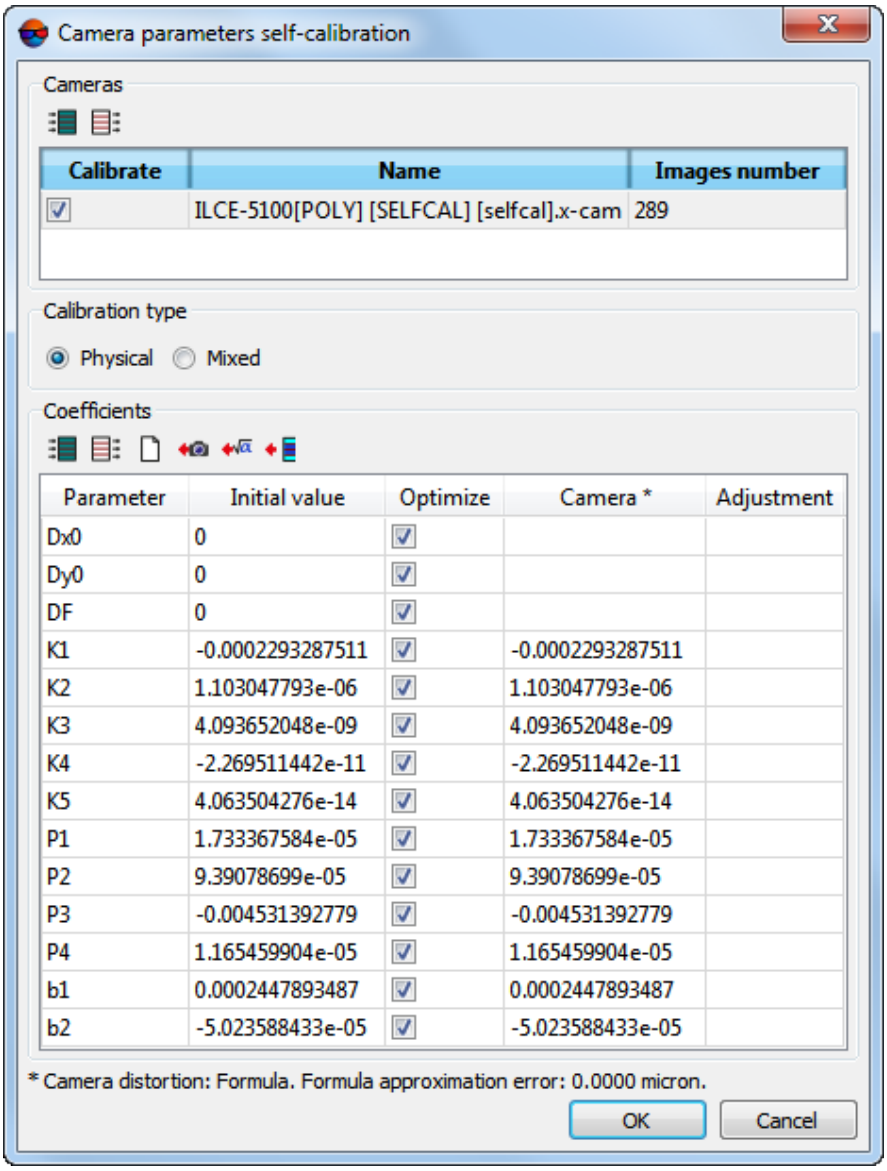


Fig. C.35. Self-calibration of camera parameters

5. Click OK. Setup adjustment parameters in the **Parameters** window and click the **Compute** button to start the adjustment procedure.

6. When the adjustment procedure is completed estimate the adjustment accuracy and results of camera self-calibration shown in the **Adjustment** column of the **Camera parameters self-calibration** window. If you are satisfied with the results pass to step # 5, otherwise, change initial values of parameters and/or set of parameters for optimization and adjust the block once more.

7. Click the **Save** button to save the adjustment results.

8. Choose **Orientation › Manage cameras**. The **Manage project cameras** window opens.

9. In the **Project cameras** list select new calibrated camera (cameras) `<...>[selfcal].x-cam` and assign it (them) to project images. Click the **Execute** button. Click OK.

   See a description of the **Manage project cameras** window in the "Aerial triangulation" User Manual.

10. Choose **Orientation › Additional › Calculate interior orientation**.

11. Choose **Orientation › Block adjustment** or click the  **Block adjustment** button on the **Aerial Triangulation** tab of the main toolbar. Click the button **Settings** on the **Adjustment** toolbar. Block will be adjusted considering new results of interior orientation.

C.6.3. Brief residuals report

The program allows to display brief residuals report, which contains RMS errors, mean absolute and maximal adjustment errors.

Brief residuals report is used for quick estimation of adjustment results without viewing detailed report.

In order to display brief residuals report, click the  button on the **Adjustment** toolbar. The **Residuals** window opens.

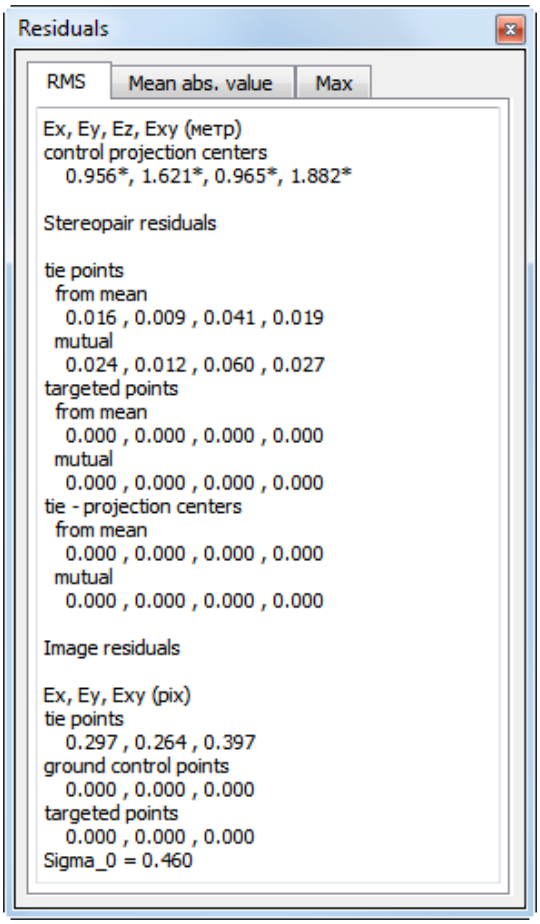


Fig. C.36. The Residuals window

In the Stereopairs mode brief report contains information about mean absolute errors on GCP and check points (in meters), about tie and targeted points, as well as about tie residuals on projection centers (*mutual* and *from mean* residuals).

If adjustment is performed by bundle adjustment procedure, the `Sigma_0` value is calculated.

The `Sigma_0` value shows how much adjustment discrepancies correspond to specified thresholds on points coordinates measurements and input GCP data (GCP and projection centers coordinates). If the thresholds were specified correctly, the `Sigma_0` has value close to 1 (±30%).

If the `Sigma_0` is much greater than 1, there are errors in points coordinates measurements, in input GCP data or because of incorrectly specified thresholds on points coordinates measurements. If the `Sigma_0` is mush less than 1, the threshold on points coordinates measurements were specified incorrectly or there are some other errors (see Section C.6.4).

C.6.4. Creating adjustment report

The program provides possibility to view full statistics of adjustment, summary information about adjustment residuals and control data.

To display adjustment report click the button on the **Block adjustment** toolbar and set the needed **Parameters** in appropriate window (see the “Block adjustment” User Manual).

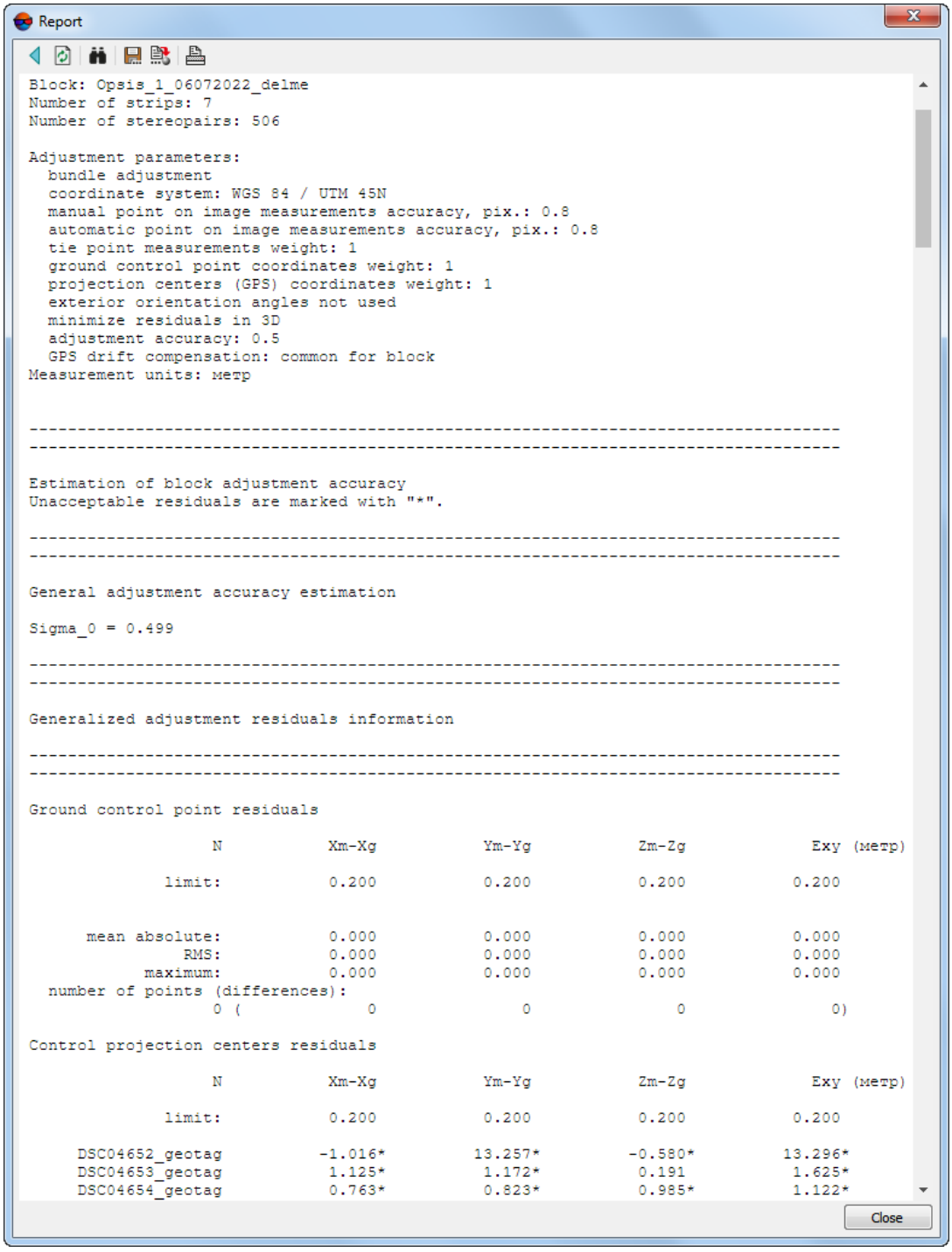


Fig. C.37. The window of adjustment report

The toolbox of the **Report** window contains the following buttons:

-  – allows to return to previous view of the report;
-  – allows to refresh the report;
-  – allows to start search in report's text (**Ctrl+F**);
-  – allows to save a report into text file;

The report also saves in the *backup* project folder automatically with the adjustment results (only for central projection images).

-  – allows to save a report into system resources;
-  – allows to print a report.

The adjustment report displays the following values:

- residual values calculated on GCP, check, tie points, and projection centers for the whole block and for each point;
- catalogue of points coordinates;
- images exterior orientation parameters;
- GPS corrections;
- corrections for exterior orientation parameters.

Report content is setup on the **Report** tab of the **Parameters** window.

Residuals and errors are displayed in the report as follows:

- *X, Y, Z* – coordinates values taken from model (from strip when using *independent strips method*, from stereopair when using *independent stereopairs method*);
- *X1, X2, Y1, Y2, Z1, Z2* – values of points coordinates on two different models;
- *Xcp, Ycp, Zcp* – values of points coordinates, averaged over all models;
- *Xg, Yg, Zg* – geodetic value of points coordinates, specified by the user for GCP and check points;
- *Exy, Ez* – average residuals of points XY and Z coordinates;
- *dX, dY, dZ, dS* – discrepancies on GCP;
- *dX, dY, dZ, dXY* – GPS corrections on projection centers.

C.6.5. Correcting errors in the positions of triangulation points

Block adjustment accuracy control includes checking the magnitude of errors in determining XY- and Z-coordinates of triangulation points. For ground control and tie points, XY- and Z-coordinate errors are displayed visually as various error vectors, and the numerical values of the errors are displayed in the adjustment report. Block adjustment accuracy control includes checking the magnitude of errors in determining XY- and Z-coordinates of triangulation points.

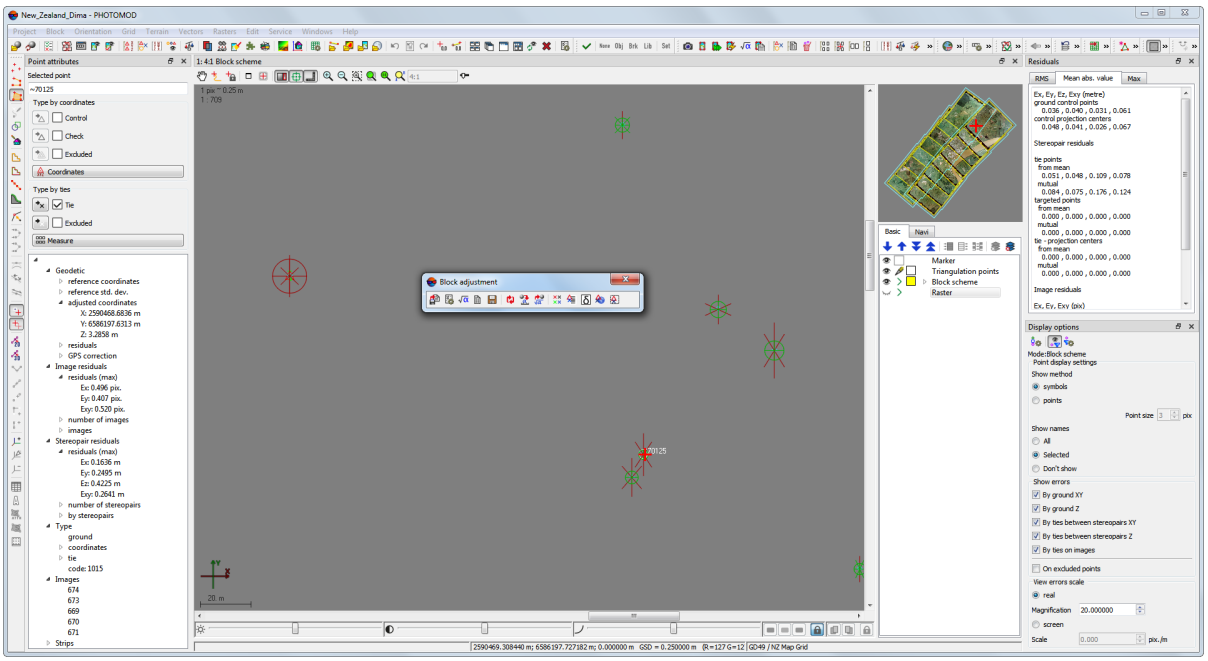


Fig. C.38. The triangulation points and residuals vectors

The triangulation points and residuals vectors displaying parameters are described in “Setup of points symbols” and “Parameters of points and error display on a block scheme” chapters in “Block adjustment” User Manual.

The causes of rough errors, as a rule, can be incorrectly specified (or, in rare cases, initially incorrect) coordinates of ground control points, as well as errors in the positioning of tie and GC points in the images. It is possible to improve unsatisfactory adjustment results, which can be partly caused by the errors described above, by editing the triangulation points and readjusting the project:

- For tie points:
  - Deleting a tie point from the project;
  - Assigning the tie point the **Excluded** type – to exclude a point from the adjustment process without deleting it from the project;
  - Correcting the point’s position in images (see below).

- For GCPs:
  - Deleting a GCP from the project;
  - Assigning the GCP the **Excluded** or **Check** point type.

    The **Check** points is not involved into block adjustment (as a GC point), but is used as a tie point. It is also can be used for the further accuracy control. The **Excluded** points type is used for temporal point exclusion from GC points list, for example, to exclude unmeasured GC points prior to block adjustment.

  - Correcting the position of the GCP in the images (for example, the GCP was incorrectly measured in the images due to an insufficiently detailed outline);
  - Correcting entered geodetic coordinates of the GCP (for example, the operator made an error when entering the coordinates of the point).

In order to change point's type or position perform the actions described below (see also full description in "The "Point attributes" window" chapter of "Block adjustment" User Manual).

Click the button on the **Block adjustment** toolbar to complete changing point's position on image.

In order to change triangulation points position on images perform the following actions:

1. On a block scheme or in a list of points select a point (or several points), which residual exceeds specified threshold value (the residuals vectors are shown by red color).

   Acceptable residuals are set in the appropriate section of the **Report** tab of the adjustment **parameters** window (see the "Block adjustment" User Manual).

2. In the **Points attributes** window click the **Measure** button. The **Points measurement** window opens (with images on which the selected points are located).

   For details see the "The "Points measurement" module" chapter in the "Aerial triangulation" User Manual.

   The list with in only selected points is displayed in the **Triangulation points** window. Only these are also displayed on images.

   Tie and ground control points are distributed into the corresponding tabs of the **Triangulation points** window (,).

   By default the filter is applied on both **Triangulation points** window tabs (display points selected on the adjustment stage). The following buttons on a toolbar of the **Triangulation points** window (in Tie points tab) allows to change the filter for points list, that are displayed in the table and on images:

- – all points;
- – points on open images;
- – common points on open images.

3. [optional] In order to display GCP residuals vectors ( ) and/or tie residuals vectors ( ) click the appropriate buttons in leftward toolbar of the **Points measurement** window;

4. Move a point manually or using correlator (see description of correlator work in the "Aerial triangulation" User Manual). Each image window in the module contains the following buttons of upper toolbox used to work with image:

- – allows to move or add selected tie or GC point to marker's position;
- – allows to move a point along tie residual vector;
- – allows to move a point along GCP residual vector;
- – allows to remove measurement results for point selected on image;

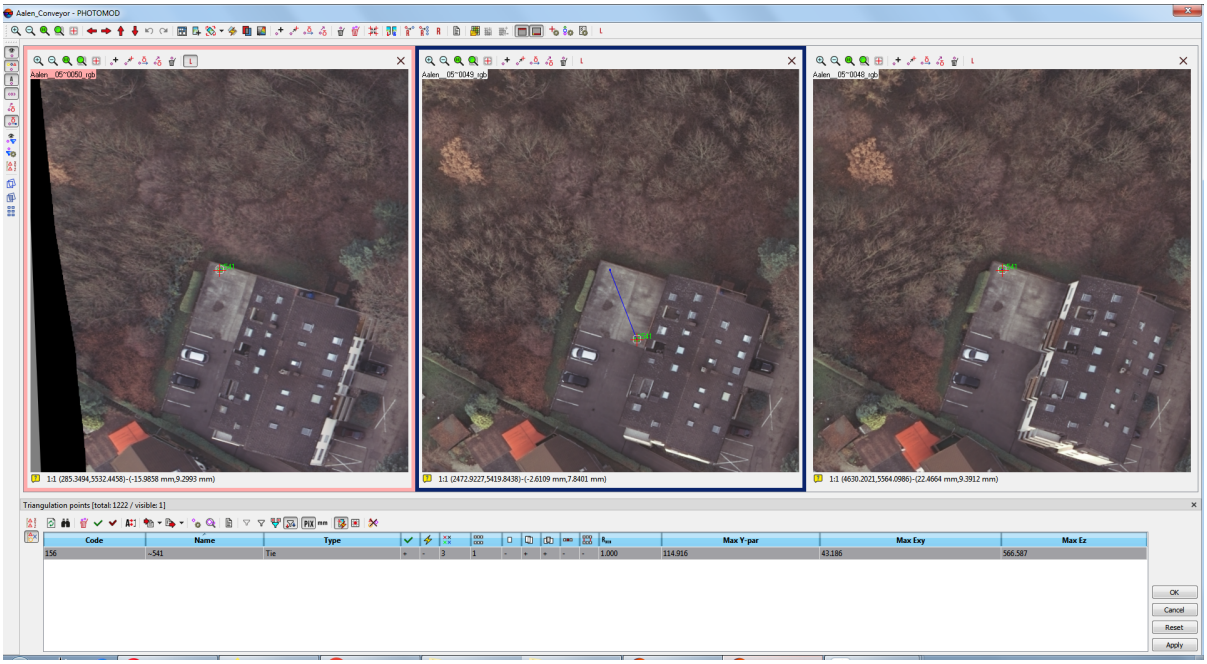


Fig. C.39. Editing tie point location. Tie residuals vector (on the central image) is colored blue

5. Click OK to exit the **Points measurement** module.

6. Click the button on the **Block adjustment** toolbar to complete changing point's position on image.

In order to use interstrip points and points non-transferred in some strip in the adjustment by *independent strips method* or *independent stereopairs method*, it is necessary to measure coordinates of the points on adjacent images inside the strip during aerial triangulation or in the **Points measurement** window (see the "Aerial triangulation" User Manual).

In order to change geodetic coordinates of points on image perform the following actions:

1. On a block scheme or in a list of points select a GC point;

2. In the **Points attributes** window click the **Coordinates** button.The **Triangulation points** window opens (on **GCP list** tab). In this window the information about selected GCP is displayed.

   Double-click of points string allows to open the **Points Measurement** window and change point's position (see above).

3. Select the GC point in **Triangulation points** window table by **left mouse button** click. Click the button in **Triangulation points** toolbar. The window **Point properties** opens:

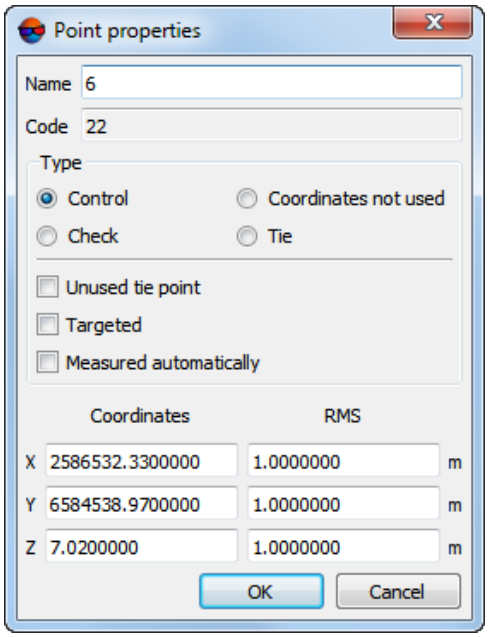


Fig. C.40. Point properties

For GC points the geodetic **Coordinates** and **RMS** of point are displayed. Specify the needed parameters and click OK to close the window.

C.7. DTM building

Refer to the "DTM Generation" User Manual for the detailed information about generation of digital elevation model (DEM).

Digital Elevation Model (DEM) – digital cartographic presentation of terrain surface both as regular grid of elevations (DEM) and as triangulated irregular network (TIN).

The following data sets (individually or in combination) are used for generation of digital terrain model:

- *Points* – point vector objects, located on relief surface;
- *Triangulation Irregular Network* (*TIN*) – one of the models of spatially coordinated data, used during designing of digital terrain model as elevation points in nodes of irregular network of triangles, that corresponds to Delaunay triangulation;
- *Contour lines* – vector lines, that connect points with the same terrain altitude;
- *Digital elevation model* – digital cartographic presentation of terrain surface as regular grid of elevation values.

The **Terrain** menu is used for working with the above mentioned data sets. Key commands are placed in the **DTM building** tab of the program's main toolbar.

C.7.1. The preparation steps

C.7.1.1. Grid creation

The program provides possibility of regular *grid* creation with specified step. A grid step is used as a frequency of nodes creation, in which vicinity spatial coordinates are calculated and points are created.

The grid could be created both for the whole images block and for any part of a block or for selected stereopair. A shape of grid borders could be rectangular or as arbitrary polygon (or several polygons).

To create a grid select **Grid › Create** (**Ctrl+N, G**) or click the  button of the main toolbar. After that a new layer *Grid* is created in the *Layer Manager*.

Area of grid creation is determined in one of the following ways:

- in order to create a grid with specified *rectangular border*, press and hold the **Shift** key, and drag by mouse a rectangular area over images block or over selected stereopair;
- in order to create a grid with *arbitrary border*, define a grid creation area in *group selection mode* () by polygon and while holding pressed the **Shift** key, specify by mouse all nodes of boundary of a grid creation area. Complete creation of arbitrary grid border by mouse double click.
- to use vector layer polygons as areas of grid creation, perform the following actions:

1. Select **Vectors › Create layer** to create vector layer without classifier or load layer with polygons which could be used as a grid boundaries.

2. Create a polygon in such a way that its boundary coincides with boundaries of area which is used for grid creation.

3. [optional] Select polygons to be used as boundaries for a grid creation, otherwise, a grid will be created considering all polygons of the layer.

4. Select **Grid › Create boundaries from vectors**. After that created grid boundary coincides with outline of created polygon.

- to create a grid for the whole images block without defining its boundaries, select **Grid › Properties**, specify its parameters and click OK. Grid is created automatically for the entire block and the grid boundary passes over external boundary of block images.

When a grid boundary is defined the system creates a grid of nodes with specified or default parameters.

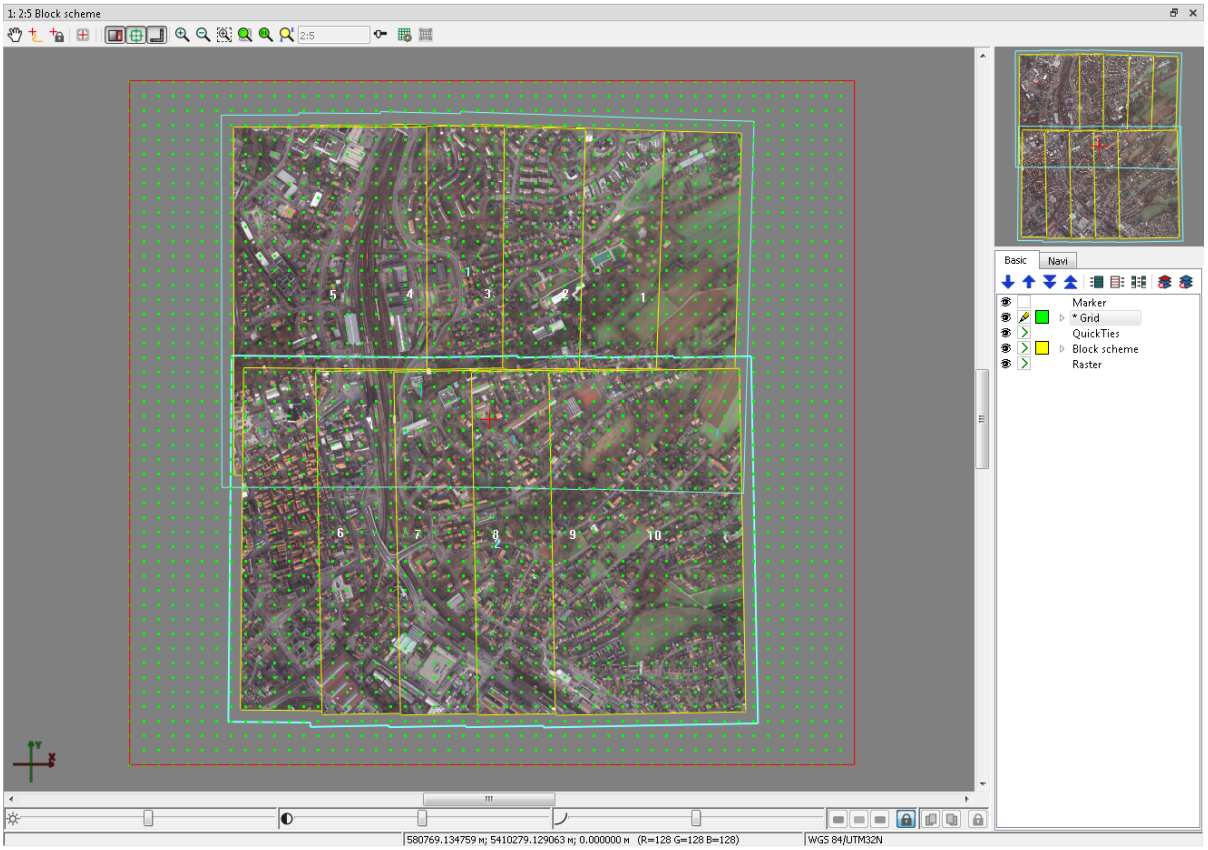


Fig. C.41. Rectangular grid for the whole images block

In order to change parameters of created grid, select **Grid › Properties**. The **Grid properties** window opens.

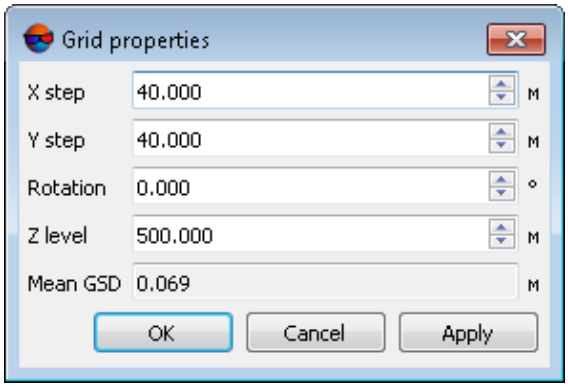


Fig. C.42. Regular grid parameters

The **Grid properties** window allows to setup the following parameters:

- **X step** and **Y step** – steps for nodes creation correspondingly by X and Y axes in meters;
- **Rotation** – angle of nodes grid rotation in degrees;

Should be specified for manual work in the pathway mode.

At that, the nodes **grid only** is rotated. Area of nodes grid creation remains in initial position.

- **Z level** – a grid elevation above relief level (in meters) for visual correct display of the grid;
- **Ground sample distance (GSD)** – shows a value of average pixel size in meters, if a project contains adjustment results or pixel size is specified for at least one image.

To display a grid with specified parameters click the **Apply** button. To change parameters click OK.

C.7.2. Compute points

This section discusses the construction of points in automatic mode (including the distributed processing mode) as well as the filtering of points that are not located on the terrain (average smoothed surface). For a detailed description of working with points, see the “Points” section of the “DTM Generation” User Manual.

Points – point vector objects, located on relief surface. Points like other vector layers are used as a base layer for TIN generation.

C.7.2.1. Automatic calculating of points

The system provides possibility to calculate points automatically using correlator in overlap areas of stereopair images by regular nodes grid.

Because of large processing data volumes in UAS projects, it is recommended to use automatic calculating of points.

All selected stereopairs should be located to adjusted part of a block (see "Block adjustment" User Manual), otherwise, points calculation is not performed or is performed incorrectly. In the first case the system displays error message, in the second one – it is not possible to match calculated points to project coordinate system.

Acquired points are used as a vector base for DEM generation. Additional possibilities of points editing allow to obtain vector base for TIN creation and DEM generation.

The principle of automatic points calculation is as follows. For each selected oriented stereopair the system performs automatic pass of all grid nodes, located in overlap area of stereopair images, and tries to calculate spatial coordinates in vicinity of each grid node using correlator.

In case of successful correlation the system performs accuracy control of each found point's coordinates, and after that the point is either added to vector layer as point object, or is excluded (see **Accuracy control with reverse stereopair** checkbox). If the system fails to calculate spatial coordinates in vicinity of some grid node, the node is skipped and the system passes to the next node.

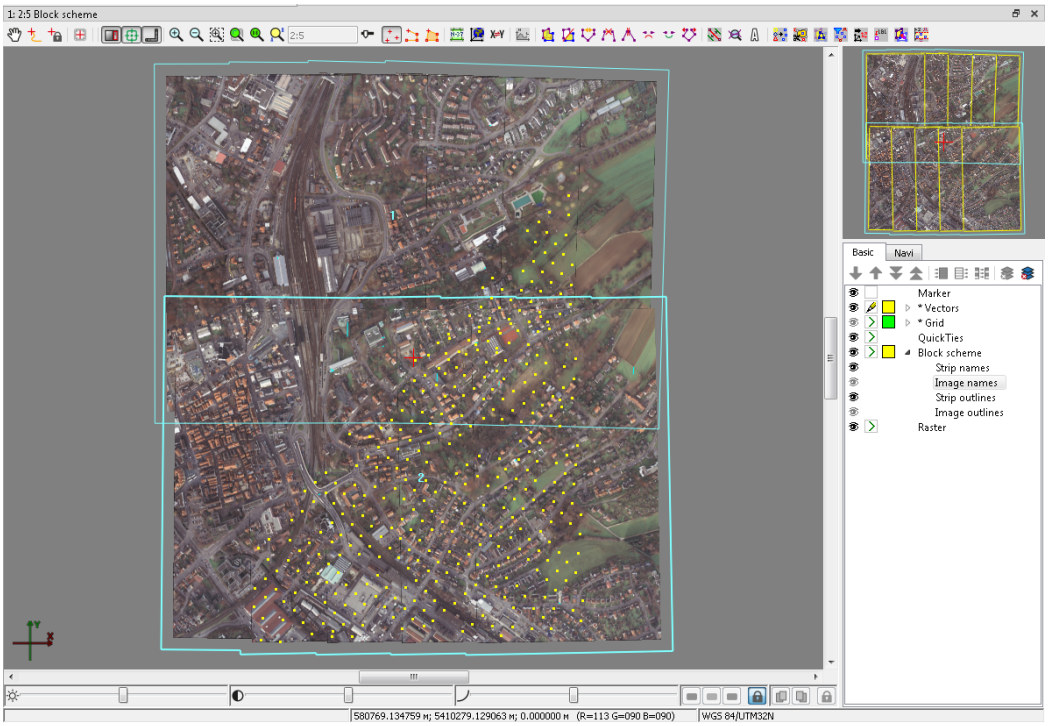


Fig. C.43. Automatic points calculation in arbitrary area

Prior to start of automatic points calculation perform the following preparatory actions:

1. Define *search area*: select block stereopairs where to perform automatic points calculation.

2. Create regular nodes grid for selected *search area*.

3. Select **Terrain › Points › Compute points automatically** or click the **Compute points** button on the **Compute DEM** tab of the main toolbar. The **Compute points** window opens.

   If a grid not was not created, it is created automatically for the entire images block and the **Grid properties** window opens, that allows to specify grid parameters.

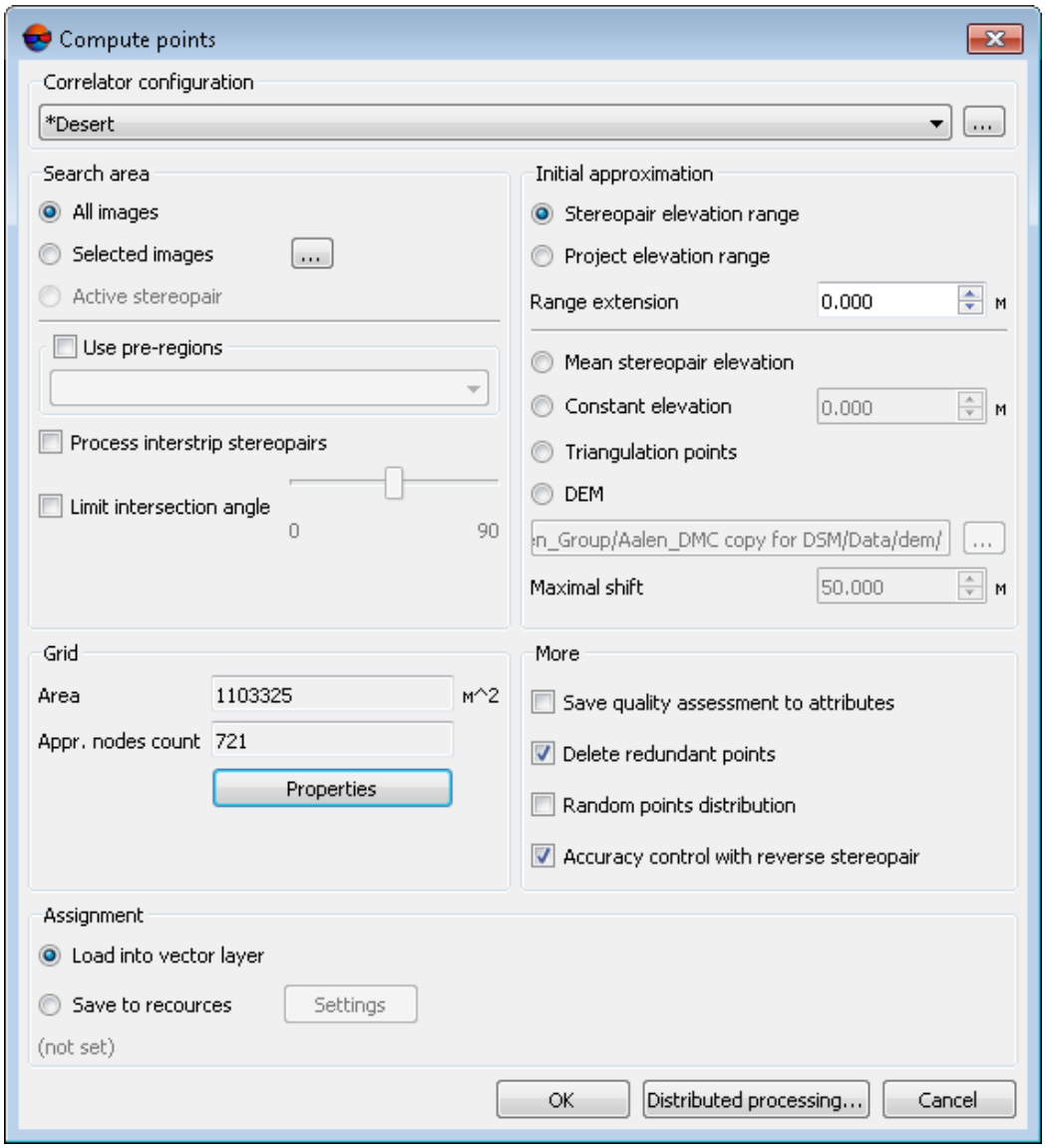


Fig. C.44. The Compute points window

4. In the **Correlator preset** section select in the list one of the following types of terrain:

- **mountainous terrain**;
- **urban area**;
- **desert**;
- **rural area**;
- **rural area 2**.

To view and edit parameters values and to setup correlator configuration the list, to create new or delete configurations from the list click the ... button (see the “Correlator configurations”, “Setup of correlator configuration”, “Additional correlator’s parameters” and “Parameters of first approximation calculation” chapters in the “General information about system” User Manual).

5. In the **Search area** section specify a search area, which will be used for automatic points calculation:
   - **All images** – allows to select all images of a block;
   - **Selected images** – allows to select block images highlighted in 2D-window;

To view and change a set of highlighted images in block 2D-window click the ... button.

   - **Active stereopair** – allows to select images of a stereopair opened in active 2D-window.

All selected stereopairs should be located on adjusted part of a block (see “Block adjustment” User Manual), otherwise, points calculation is not performed or is performed incorrectly. In the first case the system displays error message, in the second one – it is not possible to match calculated points to project coordinate system.

6. [optional] To pass each grid node only once during calculation, prior to define parameters it is necessary to create pre-regions, set the **Use pre-regions** checkbox on and select pre-regions layers in the list (see the “Creation pre-regions for a block” chapter in “DTM Generation” User Manual).

Binding of pre-regions and stereopairs is performed using `region_image_code` and `region_image_code_2` attributes.

7. [optional] Points calculation is performed by default for in-strip stereopairs. To calculate points for interstrip stereopairs, formed by selected images set the **Process interstrip stereopairs** checkbox on.

8. [optional] To filter blunders by Z on images with intersection angle of small or null value, set the **Limit intersection angle** checkbox on and define a value of minimal angle using the slider.

   The system allows to estimate the intersection angles in the stereopairs of the project (see the “The list of stereopairs for a selected image” chapter in “Creating project” User Manual and the “Creation of “stereo quality” map” chapter in “Vectorization” User Manual).

9. [optional] In the **Grid** section the **Area** of the grid layer in m^2, as well as **Appr. nodes count** in a grid are displayed in appropriate fields. To change a grid layer parameters click the **Properties** button.

   To change grid boundaries it is necessary to close the **Compute points** window.

10. In the **Initial approximation** the method of calculation of the elevation range of points with the search for the correlator:

    Initial approximation defines a value of Z coordinate of grid node, that is used to calculate initial coordinates of point on the left and right images of a stereopair.

    Selection of the most preferable method to calculate elevation range for point search depends on available data on the terrain (degree of terrain roughness, character of a region of search where automatic calculation of points takes place).

    - **Stereopair elevation range** – the elevation range is calculated for each stereopair with heights of points, previously measured in the project;
    - **Project elevation range** – the elevation range is obtained from the **Relief elevation** values in project properties or it is calculated with heights of points, previously measured in the project;
    - **Mean stereopair elevation** – mean elevation of each stereopair, calculated using exterior orientation parameters, is used as initial approximation;
    - **Fixed elevation** – input elevation value in meters to be used as initial approximation;
    - **Triangulation points** – allows to calculate the elevation range for each stereopair from a smooth model built by project triangulation points;
    - **DEM** – a value of elevation of selected DEM in point with XY coordinates of grid node is used as initial approximation.

      If grid node falls into null cell of DEM, the system uses average elevation of corresponding stereopair. To select DEM in active profile resources click the ... button.

To specify value of acceptable deviation of Z-coordinate of calculated points from initial approximation and input value in meters to the **Maximal shift** field.

11. [optional] There are the following options in the **Additional settings** section:

    - **Save quality assessment to attributes** – allows to save quality assessment of points calculation as attributes of point vector object. Quality assessments are saved to the following object attributes:

        - ’corr’ (double) – correlation coefficient, number in range from specified correlation threshold up to 1;
        - ’aq’ (double) – auto-correlation sigma;
        - ’std_dev’ (double) – standard deviation;
        - ’pass’ (int) – number of passes;
        - ’disp’ (int) – dispersion value.

      To view values of attributes of points calculation quality select the point and choose **Window › Object attributes**. The **Object attributes** window opens.

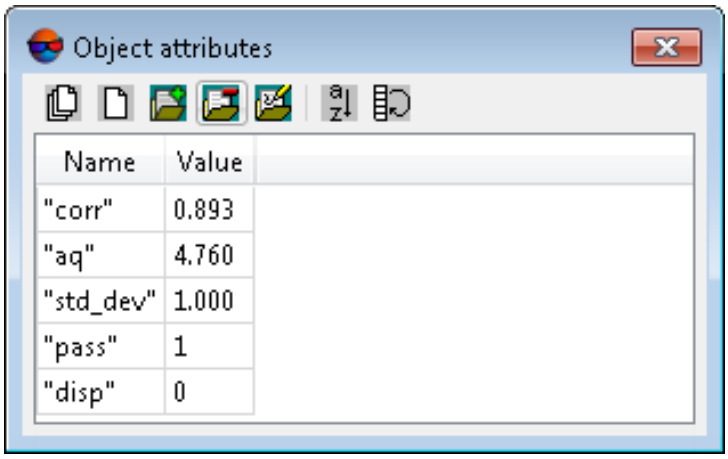


Fig. C.45. Quality assessments of calculated point

- **Delete redundant points** – allows to thin nearest points, acquired during passing the same grid nodes on different stereopairs;

Thinning is performed after completion of points calculation operation according to specified radius value, set by used correlator configuration.

When the **Save quality assessment to attributes** checkbox is on, points with minimal correlation coefficient (minimal value of ’*corr*’ attribute) are removed from several found nearest points.

- **Random points distribution** – allows distribute points randomly in area of half-size of grid cell from grid node;
- **Accuracy control with reverse stereopair** – allows to check results of found point on a reverse stereopair; if results do not the same, point removes;

12. In the **Destination** section define parameters of loading and saving of calculated points:

- **Load into vector layer** – allows to load points into active vector layer;

If there is no active vector layer, points are loaded to a new vector layer.

- **Save into resources** – allows to save points into active profile resources without loading. Click the **Settings** button to specify parameters of points saving. The **Settings for saving points** window opens.

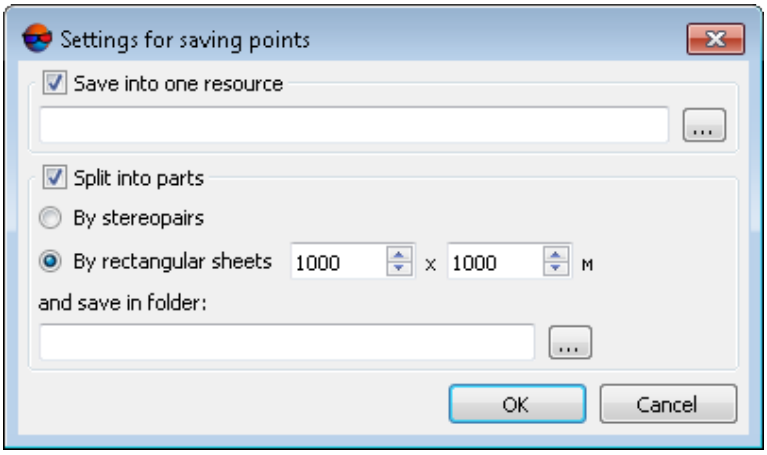


Fig. C.46. The Settings for saving points window

To save points in one vector file of active profile perform the following actions:

1. Select the **Save into one resource** option and click the ... button. The **Save** window opens.
2. Select a folder in the active profile resources and input a name of vector file in the **Resource name** field. Click OK to create a new vector file and to return to the **Settings for saving points** window.
3. The input field of the **Settings for saving points** window shows path and name of new vector file. Click OK to return to the **Compute points** window.

To save points in parts into several vector files of active profile perform the following actions:

1. Choose the **Split into parts** option.

2. Select one of the ways of points splitting – **By rectangular sheets** and input a size of rectangular area in meters or **By stereopairs**.

3. Input path to save file to the **Save in folder** field or click the ... button, to select name and path in active profile resources to save points in parts to corresponding vector files and click OK.

4. The input field of the **Settings for saving points** window shows path and name of new vector file. Click OK to return to the **Compute points** window.

13. Click OK. The points calculation operation starts.

In some cases operation of automatic points calculation takes a lot of time.

To setup points calculation in distributed mode click the Distributed processing button and specify parameters of tasks distribution.

C.7.2.2. Points calculation in distributed processing mode

To calculate points in distributed processing mode, perform the following actions:

1. Change settings and run the distributed processing server/client (see the “Distributed processing” chapter in the “General information about system” User Manual).

2. Select **Terrain › Points › Compute points automatically** or click the **Compute points** button on the **Compute DEM** tab of the main toolbar. The **Compute points** window opens.

If a grid not was not created, it is created automatically for the entire images block and the **Grid properties** window opens, that allows to specify grid parameters.

3. Setup parameters of automatic points calculation.

4. Click the **Distributed processing** button. The **Compute points Distributed Processing** window opens.

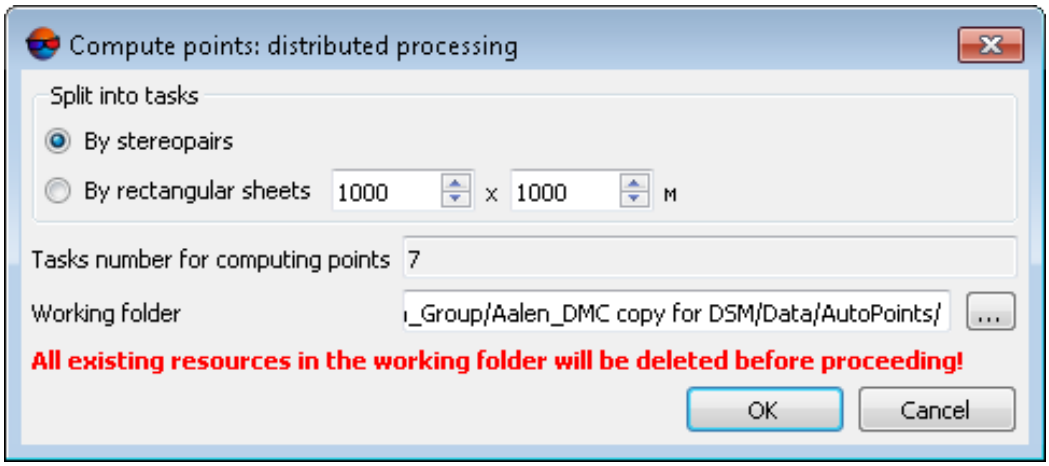


Fig. C.47. Parameters of distributed processing of points calculation

5. In the **Split into tasks** section select a way of splitting of processing operation into tasks:

   - **By stereopairs** – allows to process each stereopair in a separate task;

     Since tasks are executed independently from each other, the system doesn't remove points, falling into the same grid nodes from different stereopairs (for example, during filtering of nearest points).

   - **By rectangular sheets** – it is necessary to specify in the fields a size of rectangles, into which the entire search area is split (in meters). Filtering of nearest points is not performed in this case. The result of each task processing is saved to the file with name like `Sheet_X_Y.x-data`.

     Rectangle, circumscribing grid boundaries in project coordinates system, is split by rectangular sheets of a size specified in the fields. Each sheet includes images parts from all stereopairs, into which it falls.

   If stereopairs are processed in separate tasks, after that it is necessary to perform manual data correction. Calculated points are saved in resources, named by stereopairs names.

   The **Tasks number for computing points** field displays calculated number of tasks depending on selected way of workflow split and/or size of specified areas.

   By default По умолчанию задается вложенная папка в папке *\Data* текущего проекта.

   It is not possible to define number of tasks manually.

6. In the **Working folder** section click the ... button and select **empty** folder in active profile resources to save output DEM.

Before operation is performed all data from selected folder is removed. It is strongly not recommended to specify a project folder as a working folder, especially when there are no saved project backups.

7. Click OK. Distributed processing tasks are created and the system shows a message about number of created tasks.

When using distributed points calculation, the result of each task is saved to separate file with `*.tsk` extension in specified **Working folder** of a project.

C.7.2.3. Buildings and vegetation filter

The system provides deletion, correction or recognizing points which have fallen on buildings, trees, cars, in pits during automatic points calculation, and during filtering of accident surges. For that filter of buildings and vegetation id used. After its work only points that describe terrain relief are remain in the system.

In this document all points not lying on the relief surface (average smoothed terrain surface), are called *surges*.

Filter of buildings and vegetation allows to apply step-by-step points filtering using particular script, i.e. to apply points filtering in multiple passes with different parameter sets.

It is possible to develop customized script of filter workflow or to use default script. Creating script of buildings and vegetation filtering means specifying of passes set, order of their passing and setup the parameters for each pass.

Step-by-step filtering allows to achieve optimal results of objects filtering on any type of terrain to acquire points, that correctly describe terrain relief, and that could be used to generate high quality DEM.

By default, the system uses standard script that includes three passes in the following order:

1. **Basic** – main pass with parameters configured for filtering points not lying on the relief surface (surges). On this step the system rejects majority of points on buildings, and also correlators blunders (sharp spikes).

2. **Additional** – additional pass with parameters configured for searching surges, missed during basic step.

3. **Detailed** (off by default) – the pass with parameters configured for filtering points located on low objects, such as small buildings, cars, and so on.

After filtering the system may perform the following actions with points found (surges):

- rejecting surges from initial points layer;
- correction of surges found on initial points layer, that means editing of surges Z-coordinates;
- search for surges to analyse them with saving of found points to new vector layers without changing initial points layer.

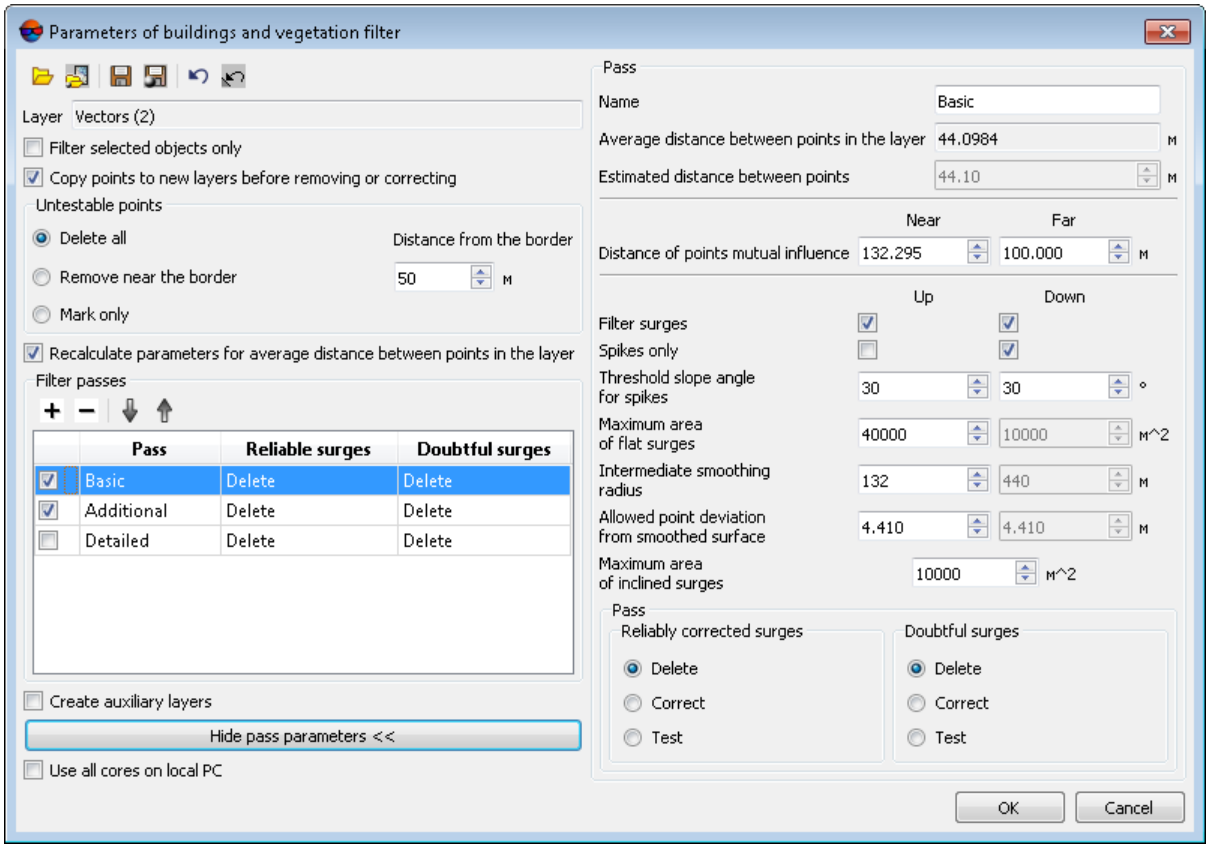


Fig. C.48. Parameters of buildings and vegetation filter

The **Buildings and vegetation filter** window allows to specify filtering parameters and setup a number of filter passes. The **Layer** field displays a name of active vector layer. The window contains a standard toolbar.

Buttons	Function
	to load a script of filter work from `*.x-filter` resource located out of active profile resources
	to load a script of filter work from `*.x-filter` resource located in active profile resources
	to save a current script of filter work to `*.x-filter` resource located out of active profile resources

Buttons	Function
	to save a current script of filter work to `*.x-filter` resource located in active profile resources
	to cancel all changes made to a script
	to go back to a standard script that includes two passes with default settings (regardless of which script has been loaded)

To filter objects on surface perform the following actions:

1. Load points for filtration or make active a points layer.

   To avoid loss of data, it is recommended to use a copy of initial layer during filtering.

   To create a copy select **Terrain › Points › Save as** and specify new name of points file.

   If a layer contains linear or area vector objects the filter may not work properly.

2. Select **Terrain › Points › Filter › Buildings and vegetation filter** or click the **Buildings and vegetation filter** button on the **Compute DEM** tab of the main toolbar. The **Buildings and vegetation filter** window opens.

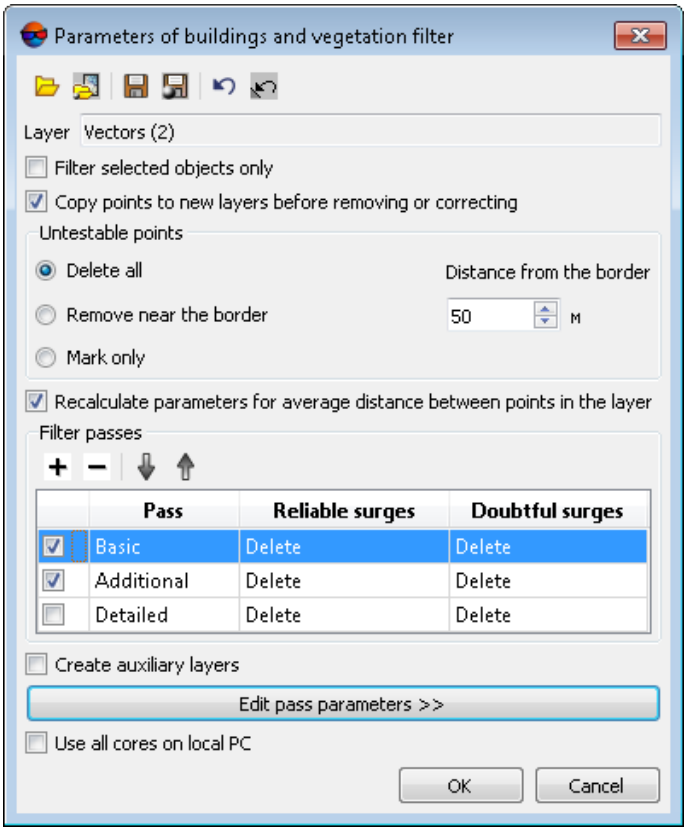


Fig. C.49. Parameters of buildings and vegetation filter

3. [optional] In order to use filtering for selected objects (points group), set the **Filter selected objects only** checkbox on. Otherwise, filter is applied to all points of the layer.

   To filter a group of points, select it by mouse in 2D-window prior to start configuring the **buildings and vegetation filter**.

   When using filter for selected objects only, it is not recommended to set the **Remove near the border** option in the **Untestable points** section.

4. [optional] To increase the filtering operation performance set the **Copy points to new layers before removing or correcting** checkbox off. By default, the system saves removed or corrected points in new vector layers used for analysis of filtering results. These layers could be used to restore basic points layer (using operation of layers merging), if the copy of initial points layer was not created prior to filtering start.

   If the **Copy points to new layers before removing or correcting** checkbox is off, it is highly not recommended to apply filtering to initial points layer, since it is impossible to restore the initial layer after filtering.

5. In the **Untestable points** section specify one of the following actions with untestable points found after filtering:

   Some points can be *untestable* – the points, in which vicinity there is not enough 'neighbour' points. For example, points located on the edge of the whole layer or in areas of low density of points (fields, forests, water bodies).

   - **Remove all** – allows to remove all found untested points;
   - **Remove near the border** – allows to set a value of distance in meters from the edge of the whole vector layer, at which all untestable points will be removed. The **Distance from the border** field is used to do this;
   - **Mark only** – allows to save untested points to separate vector layer for further analysis.

6. [optional] Values of average distance between points in active layer calculates automatically. Also the **estimated distance between points** is equal to average distance by default. Set the **Recalculate parameters for average distance between points in the layer** checkbox off to set the **estimated distance between points** manually.

7. Specify a number of filter passes:

   - by default it is used two passes with standard parameters – **Basic** and **Additional**. Set off checkboxes to change set of passes;

- the + button allows to add a new pass of filter;
- the − button allows to remove selected pass of filter;
- the and buttons allow to move selected pass up/down.

The **Filter passes** section contains a table of passes and buttons used to change set of passes and their order with the following columns:

- **Pass** – displays pass name and checkbox used to include a pass to script or to exclude it from filter workflow script;
- **Reliable surges** – displays action to be applied to well-defined surges found, that are completely comply with specified filtering criteria;
- **Doubtful surges** – displays action to be applied to ambiguous surges found, in which vicinity there is no data enough for analysis.

It is necessary to select an action to be applied to reliable and doubtful surges during configuring of pass parameters.

8. Select a pass name and click the **Edit pass parameters> >** button. The **Pass** section is opened and allows to display and configure the following parameters of filter pass:

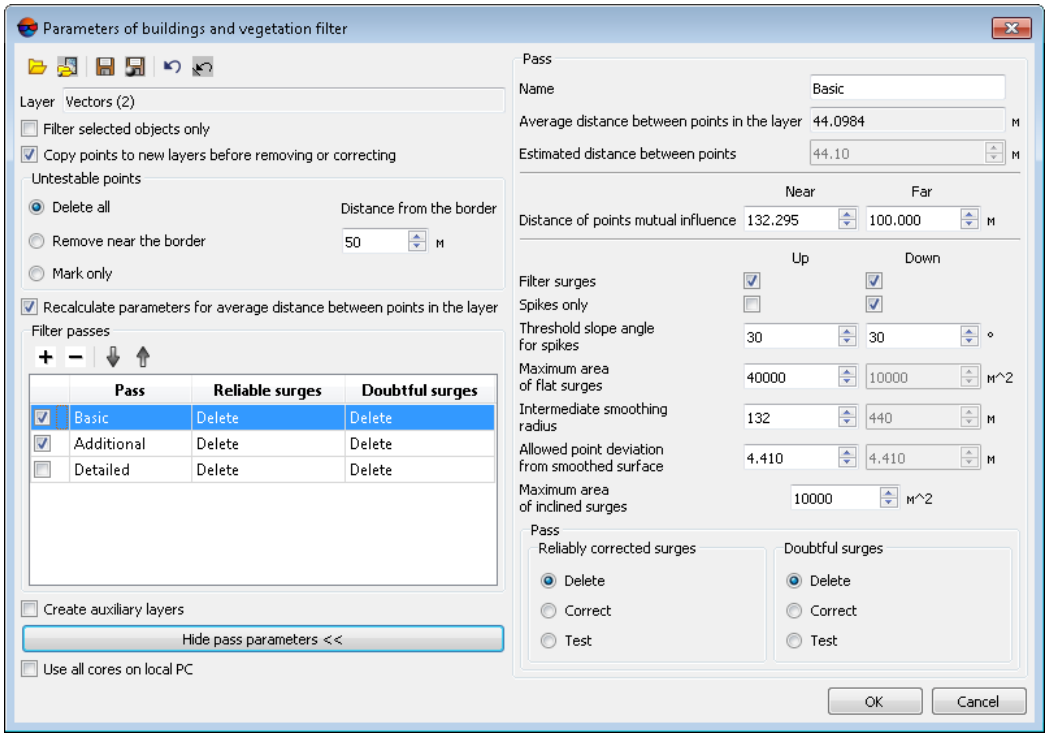


Fig. C.50. Parameters of buildings and vegetation filter

- [optional] to change name of pass, input a new one in the **Name** field;
- the **Average distance between points in the layer** (in meters) and **Estimated distance between points** (in meters) after filtering are displayed in fields;
- specify **Near** and **Far distance of mutual points influence** in meters to define the radius of the circle, in which the values of points are analysed for errors;

Near distance of points mutual influence – is the average radius of a circle where the points are located that are attributed to the same flat surface towering above the terrain (e.g. a flat roof).

Far distance of points mutual influence – is the average distance from points attributed to the flat surface towering above the terrain where the points attributed to the terrain are located for sure.

It is recommended to set the near distance (minimum circle radius) value as 3-5 values of **Average distance between points in the layer** parameter.

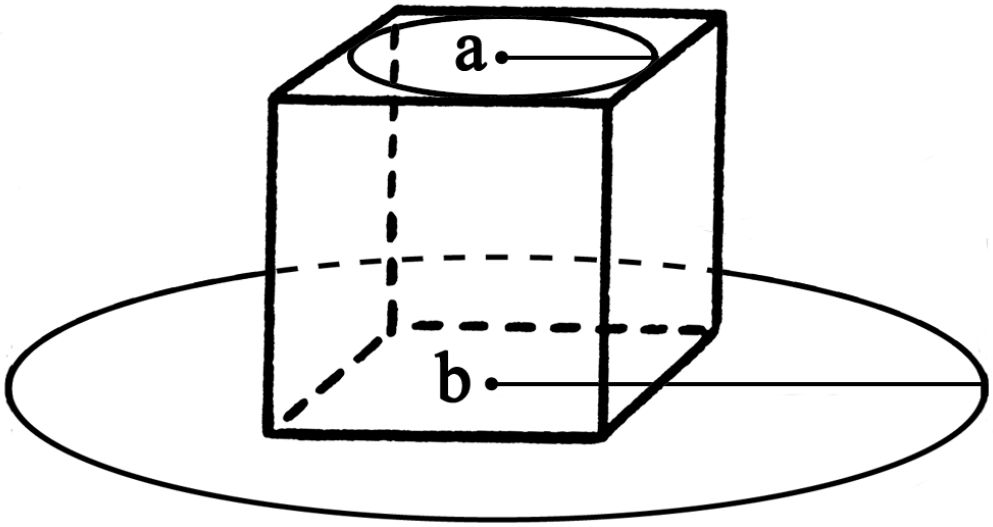


Fig. C.51. Near distance of points mutual influence (a) and far distance of points mutual influence (b)

- specify the following filtering parameters **Up** (above surface) and/or **Down** (below surface):
  - **Filter surges** – allows to select points to be filtered: points on surface (high objects) and/or points below surface (pits);

For filter operating it is necessary to set at least one checkbox on.

- **Spikes only** – is used to reject just sharp spikes above/below surface, that are defined by values of the **Threshold slope angle for spikes** and **Distance of points** parameters. The rest of parameters are not considered;

In an angle between three points exceeds a value of the **Threshold slope angle for spikes** parameter, then the surge is called *spike*.

- **Threshold slope angle for spikes** – allows to specify angle of slope in relation to selected measuring plane (above and/or below surface) to define sharp spikes;

- **Maximum area of flat surges** – allows to define maximal area of plane surges (above/below surface) – points groups, that form smooth surfaces and are distant from some plane of neighbour points. Generally, the points are located on buildings roofs, and are lying on the same plane;

The filter is not applied to plane surface, which area exceeds the specified value.

- **Intermediate smoothing radius** – allows to specify a sphere radius (above/below surface), that defines a level of intermediate smoothing of surface;

- **Allowed point deviation from from smoothed surface** – allows to specify criterion, according which the filter is applied to all points with elevations that differ from smoothed surface more then on specified value;

- **Maximum area of inclined surges** – allows to define maximal area of inclined surges (above/below surface) – points groups, that form inclined surfaces on the slopes.

The  button allows to go back to a standard script that includes two passes with all default settings (regardless of which script has been loaded).

9. In the **Actions** section specify actions for reliable and doubtful surges found after any step of filtering:

   - **Delete** – deleting of points found from base points layer;
   - **Correct** – editing in base layer Z-coordinates of points found;
   - **Check** – saving of found points in a new layer without change of base layer.

Reliable corrected surges – found well-defined surges that satisfy completely to all specified filtering criteria.

Doubtful surges – ambiguous surges in which vicinity there is no data enough to analyse filtering operation possibility.

If the **Delete** or **Correct** option is selected, the system allows to edit base points layer.

In order to save deleted or corrected points to new layers it is recommended to set the **Copy points to new layers before removing or correcting** checkbox on.

If the **Check** option is selected the initial points layer remains unchanged, and found reliable or doubtful surges are copied to a new layer *Surges*.

10. [optional] To use all cores of workstation CPU for calculation operation, the **Use all available CPU cores** checkbox is set on in the system by default. It is necessary to set the checkbox off to use only one core.

11. Click OK. The system starts DEM filtering operation. When the filtering operation is completed the system displays information message about number of used basic points and filtered points.

There are the following recommendations concerning use of buildings and vegetation filter:

- it is recommended to use step-by-step filtering (in multiple passes with different parameter sets);

- the passes should be formed in ascending order of the following parameters: distances of mutual influence, radius of intermediate smoothing, and decreasing of allowed deviation of point from smoothed surface;

- value of intermediate smoothing radius directly describes terrain relief features, that it why it is necessary to specify the radius value not exceeding 1500 meters;

- it is recommended to perform preliminary analysis of maximal area of flat surges on given territory (to measure it in a stereopair window) and then to compare it with area, calculated automatically for passes. If the measured area is bigger than calculated one, it is necessary to increase it;

- it is highly not recommended to specify a distance of mutual influence of points exceeding a step between points in more than 20 times, since it leads to considerable slowdown of filtering operation.

C.7.3. Build TIN

Refer to the “Triangulation irregular network (TIN)” chapter of the “DTM Generation” User Manual for the detailed information about TIN generation.

Triangulation Irregular Network (*TIN*) – one of the models of spatially coordinated data, used during designing of digital terrain model as elevation points in nodes of irregular network of triangles, that corresponds to Delaunay triangulation;

Delaunay triangulation – is a triangle polygonal network, formed on a set of discretely placed points connected to each other by disjoint straight line segments in such a way, that a circle circumscribed around each triangle does not contain points of initial set inside.

To perform TIN creation the system provides the following workflow:

1. [optional] Pre-regions building (see the “Creation pre-regions for a block” chapter of the “DTM Generation” User Manual);
2. Defining of set of base vector layers (see the “Loading base layers for TIN creation chapter of the “DTM Generation” User Manual);
3. TIN borders creation (see the “TIN borders creation” chapter of the “DTM Generation” User Manual);
4. Building TIN.

C.7.3.1. Building TIN

Perform the following actions to build TIN:

1. Choose the **Terrain › TIN › Build** (**Ctrl+N, T**) or click the **Build TIN** button on the **Compute DEM** tab of the main toolbar. The **Build TIN** window opens.

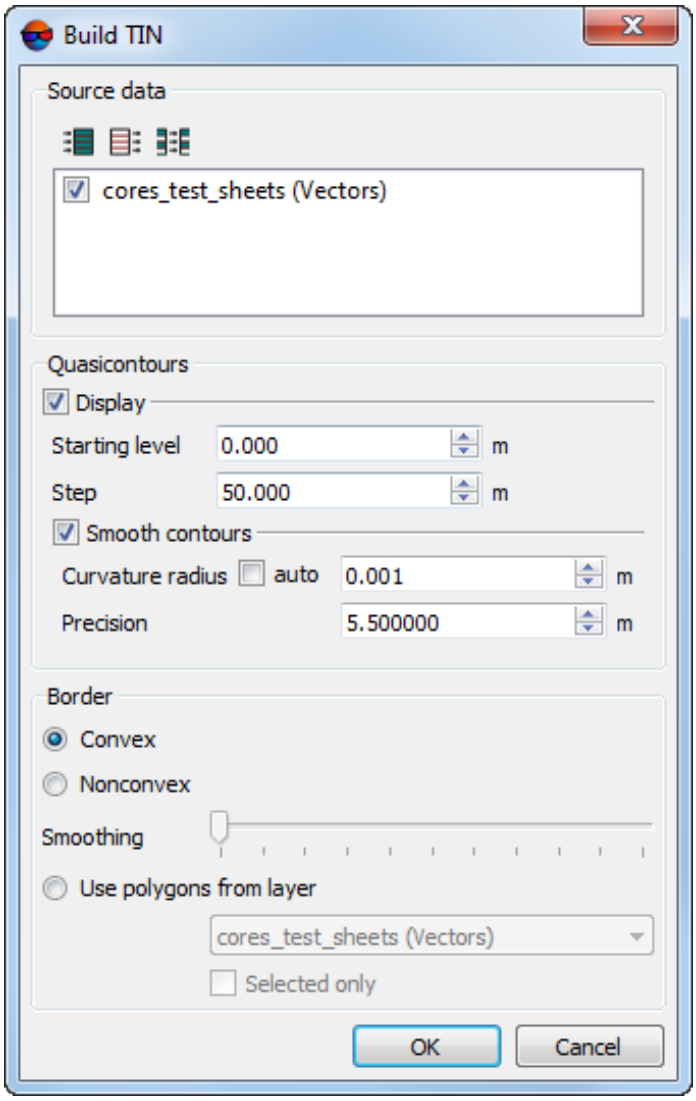


Fig. C.52. The Create TIN window

2. [optional] By default in the **Source data** section set on checkboxes for all opened layers to use them in TIN creation. Set the checkboxes off, if some layers opened are not used for TIN creation.

   To select all available layers click the button, to unselect all layers – click the button. To invert selecting layers it is used the button.

3. [optional] In the **Quasicontours** section the **Display** checkbox on and in the **Starting level** filed specify minimal Z level (Zmin), from which to start quasi-contours creation.

4. [optional] Specify the **Interval** of quasi-contours creation in meters.

5. [optional] In order to create quasi-contours as smoothed curves, set the **Smooth contours** checkbox on and specify the smoothing parameters.

   - Input the **Curvature radius** for smoothing curve.

     Set the **auto** checkbox to calculate the **Curvature radius** for smoothing curve automatically.

   - Specify in the input field the **Precision** parameter – maximal distance from polyline segment to a curve in the area between the two closest vertices.

6. In the **Border** section set on the following:

   - **Convex border** – a border is created with connecting of outer boundary points of selected layer in such a way that TIN has the most smooth border;

     It is recommended to create convex border, when some part of vector objects does not cover all area of TIN creation (for example, when there are lakes and rivers on large scale images).

   - **Nonconvex border** – only closest border points of a layer are connected during border creation.

     The **Smoothing** slider position allows to specify distance between border points, and if the distance between points is less than the specified, the border will be passed through them. In the leftmost position of the slider point are connected sequentially, and in the rightmost one the border looks like convex.

     It is recommended to place the **Smoothing** slider in the middle and smoothly move it to the left to get the best results.

   - **Use polygons from layer** – to create a TIN with convex border in limits of area of polygon(s) selected in the layers list. Select from the list a layer containing polygons, that will be used as a border. In order to define a border of TIN creation area, just from selected polygons, set the **Selected only** checkbox on.

7. Click OK. The TIN is created in the new *TIN* layer.

C.7.4. Build DEM

Refer to the “Digital Elevation Model” chapter of the “DTM Generation” User Manual for the detailed information about DEM generation.

DEM (Digital Elevation Model) – is digital cartographic representation of the earth’s surface in the form of a regular grid of height values.

A source data for DEM creation is the following data, taken individually or in combination:

- TIN (Triangulated Irregular Network);
- regular or irregular points (point objects);
- vector objects (see the *Vector objects creation* chapter in the "Vectorization" User Manual).

C.7.4.1. DEM creation by TIN

The system provides possibility to create DEM using irregular spatial triangles network (TIN). Perform the following actions to do this:

1. Create or load source TIN.
2. Choose **Terrain › DEM › Build DEM › From TIN** (**Ctrl+N, D**) or click the **Build from TIN** button on the **Compute DEM** tab of the main toolbar. The **DEM by TIN creation parameters** window opens.

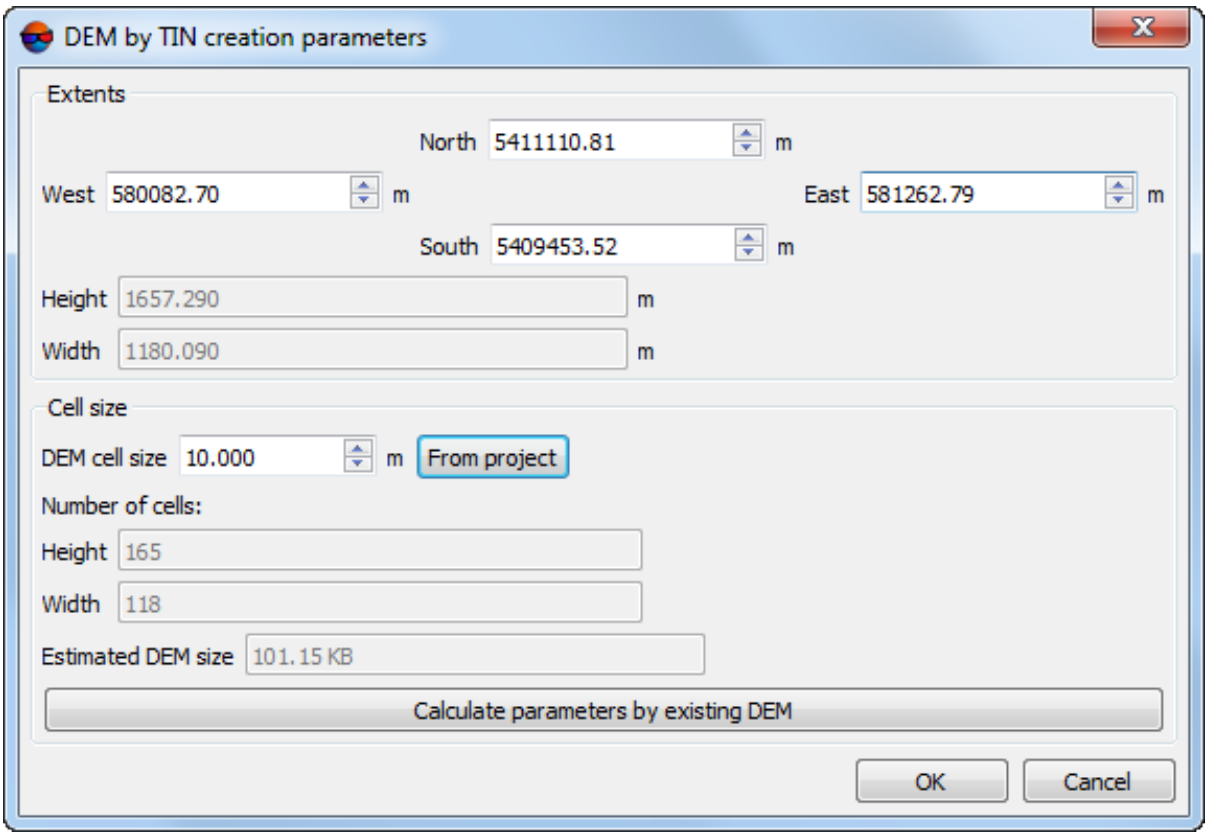


Fig. C.53. DEM by TIN creation parameters

3. Specify DEM borders in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters.

Default values are coordinates of rectangle corners, which generate TIN creation area.

4. In the **Cell size** specify the **DEM cell size** in meters to define cell size of output DEM. The **Cells number**, calculated considering specified cell size, displays in the **Height** and **Width** fields. Also the **Approximately DEM size** in megabytes displays in the appropriate field.

DEM cell size should be comparable to average distance between points of base TIN layer. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

Click the **From project** button to setup DEM cell size that equals to average GSD value in the project.

5. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

6. Click OK. Specify DEM file name, define a folder in active profile resources and click **Save**. The system starts DEM creation operation. After that the system creates new layer in the *Manager* and shows a message about operation successful or unsuccessful completion.

When marker is moving over DEM, the status panel in 2D-window displays XYZ-coordinates of DEM points.

The system allows to recreate DEM after changing of TIN base layer. To do this use the **Terrain › DEM › Rebuild by TIN** menu item.

C.7.4.2. Dense DEM generation using SGM method

The system provides possibility of generation of dense digital elevation model (DTM), which cell size coordinates to one pixel of image, using SGM (Semi-Global Matching) method.

DEM creation using dense model is executed for adjusted images block only (see "Block adjustment" User Guide). Otherwise the DEM generation is not performed or is performed incorrectly.

For dense DEM generation using SGM method the following output data is provided:

- [optional] DEM generation for a predetermined area;
  - [optional] **True Ortho** creation for an area limited by DEM boundaries.
- [optional] **Point cloud (LAS)** generation for an area limited by DEM boundaries.

For dense DEM generation using SGM method the system allows the following workflow:

1. DEM generation for a predetermined area;

2. Main processing: applying buildings and vegetation filters to created DEM and/or performing empty cells restoring;

3. [optional] Additional processing — median and/or smoothing filtering.

If intermediate data is not deleted after operation completing, the system allows to run individual stages of the processing (filtering and smoothing) without re-calculation of the source DEM.

To generate dense digital elevation model using SGM method perform the following actions:

1. Specify a grid to define DEM creation area.

Since the coordinates calculation is performed in each pixel of the selected area, the grid nodes are not considered. The grid is used just as a border of DEM creation area. A shape of grid borders could be rectangular or as arbitrary polygon (or several polygons).

2. Select **Terrain › DEM › Build DEM › Dense DSM (SGM method)...** or click the  **Dense model** button on the **Compute DEM** tab of the main toolbar. The **SGM parameters** window opens.

If a grid not was not created in advance, the **Grid properties** window opens, that allows to create a grid for the entire images block.

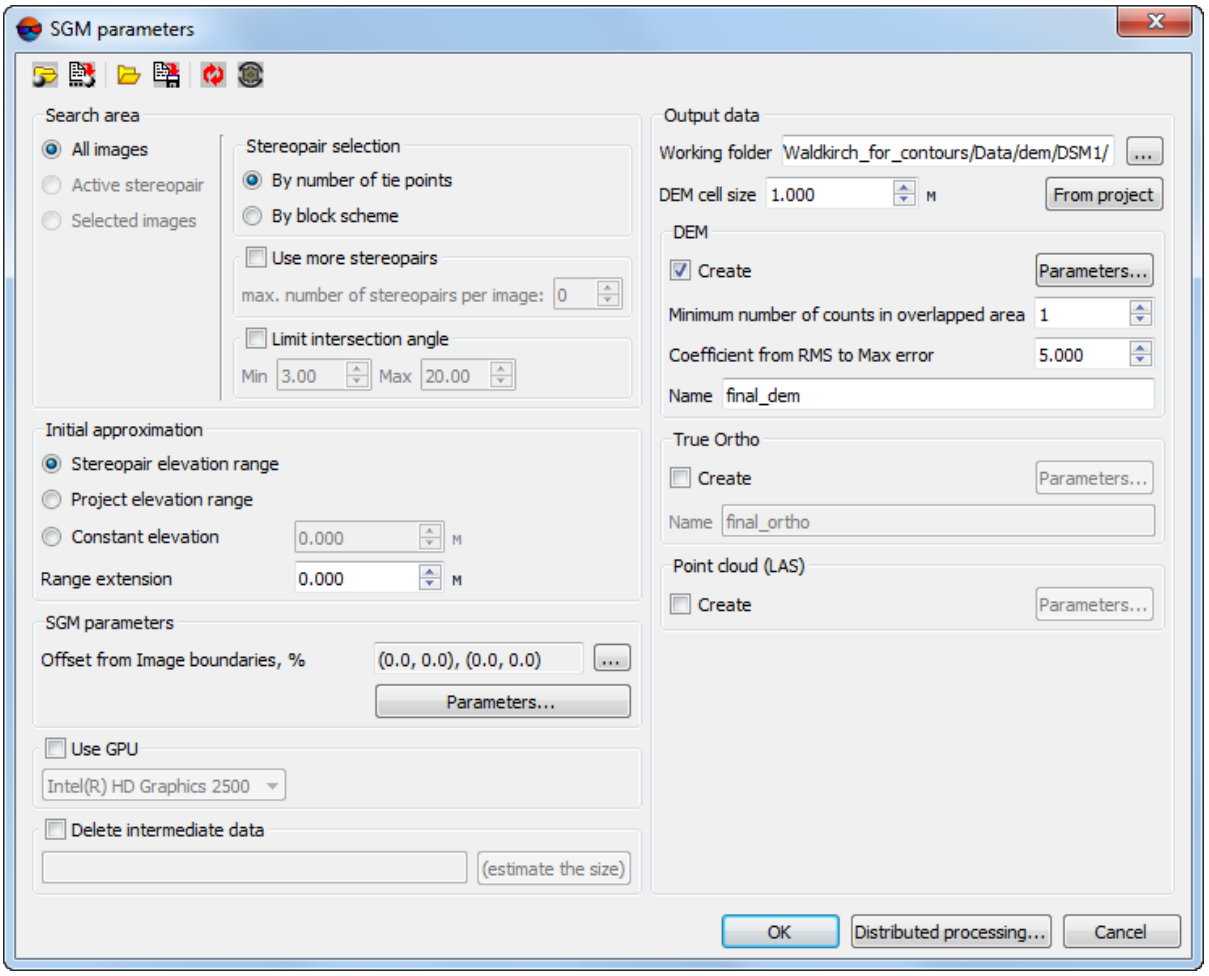


Fig. C.54. SGM parameters

The toolbar in the upper part of the window is intended to save/load parameters settings of DEM creation and contains the following buttons:

- – allows to load parameter's settings saved previously to active profile resources;
- – allows to save all parameter's settings to `*.x-ini` file located in active profile resources;
- – allows to load parameter's settings from *`*.x-ini`* file located in a folder of file system;
- – allows to save parameter's settings to `*.x-ini` file located in a folder of file system;
- – allows to restore the last-session's settings;
- – allows to restore default parameter's settings.

3. Define the **Search area** in the appropriate section:

   - **All images** – allows to select all images of a block;
   - **Selected images** – allows to select block images highlighted in 2D-window;
   - **Active stereopair** – allows to select images of a stereopair opened in active 2D-window.

   All selected stereopairs should be located on adjusted part of a block (see "Block adjustment" User Guide). Otherwise, the calculation is not performed or is performed incorrectly. In the first case the system displays error message, in the second one – it is not possible to match calculated points to project coordinate system.

4. Specify the settings that determine which **stereopair selection method** will be carried out:

   - **By number of tie points** – by default, during processing, one stereopair is selected for each of the images, based on the maximum number of common tie points. The stereopair search is carried out over the entire block of images (both in-strip and among images located on neighboring strips);

     ○ [optional] To **use more stereopairs**, set the appropriate checkbox. The search for additional stereopairs is also carried out among all the images. It should be noted, that the number of potential stereopairs for each image (in particular, taking into account inter-strip pairs) can be quite large. The system allows the user to limit the **max. number of stereopairs per image** to no more than the specified one, in the appropriate field.

     Using additional stereopairs allows for compensating for insufficient overlaps between images, which, under certain circumstances, can lead to errors in the construction of output products, especially at block boundaries.

     Inter-strip stereopair use may be necessary when processing **Vision Map** projects (depending on source data features).

   - **By block scheme** – one stereopair is selected for each of the images according to the arrangement of images in the **Block editor** window (see "Images block forming" of the "Creating project" User Manual). The creation of in-strip stereopairs only is supposed (if the user has decided not to **use more stereopairs**).

     Such an approach implies the correct arrangement of images in the block scheme and a sufficiently high image overlap, which is not always feasible when processing UAS data (in this case, the formation of stereopairs **by block scheme** is not recommended). The formation of stereopairs **by block scheme** is justified in the lack (or insufficient number) of tie point measurements

Note, that in the latter case DEM creation using dense model can be performed correctly for adjusted projects only. In turn, the steps of measuring tie and ground control point coordinates can be excluded from project processing only if quite accurate exterior orientation parameters of block's images are available what makes it possible to use these data for the project adjustment (see "Import of exterior orientation parameters" of the "Aerial triangulation" User Manual).

- [optional] To **use more stereopairs**, set the appropriate checkbox. It should be noted, that the number of potential stereopairs for each image (in particular, taking into account inter-strip pairs) can be quite large. The system allows the user to limit the **max. number of stereopairs per image** to no more than the specified one, in the appropriate field.

  The search for extra stereopairs is carried out over the entire block of images (including among images located on neighboring strips).

5. [optional] set the **Limit intersection angle** checkbox on to filter blunders by Z on images with intersection angle of small or null value, and define a value of minimal and maximal angles;

   The system allows to estimate the intersection angles in the stereopairs of the project (see the "The list of stereopairs for a selected image" chapter in "Creating project" User Manual and the "Creation of "stereo quality" map" chapter in "Vectorization" User Manual).

6. In the **Initial approximation** section select a method of heights range calculation for DEM generation:

   - **Stereopair heights range** – heights range is calculated for each stereopair individually using heights of project tie points measured earlier;
   - **Project elevation range** – heights range is taken from the **Relief elevation** value in project properties or is calculated using heights of project points measured earlier;
   - **Constant elevation** – input a height value in meters to be used as initial approximation;

   The **Range extension** field allows to increase height range by a predetermined value to create a DEM. If the **Constant elevation** mode is on the option specifies working range of heights (search area).

7. In order not to use boundary regions of imagery while DEM (LAS, true ortho) creation, set the **Offset from image boundaries, %** by clicking the ... button in the **SGM parameters** section. The **Image margin** window opens. Set image margins in percent which are not to be involved in processing;

   It may be necessary so that only useful part of an image was processed (e.g. while processing projects with analog imagery as source data).

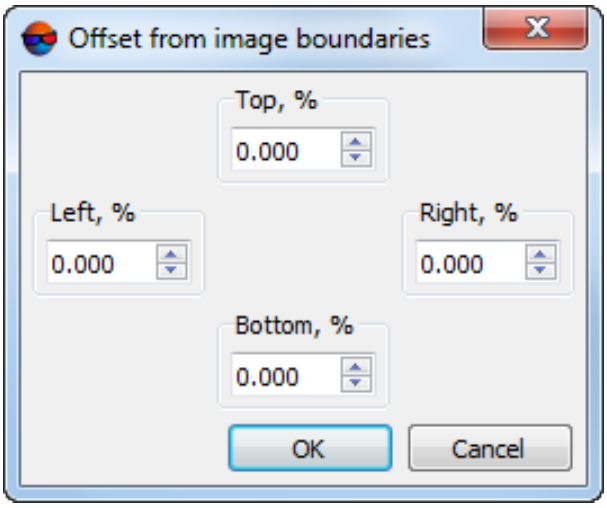


Fig. C.55. The Image margin window

Fig. C.56. Image margin

8. For more detailed settings of SGM cost calculation click **Additional...**. The **SGM parameters** window opens.

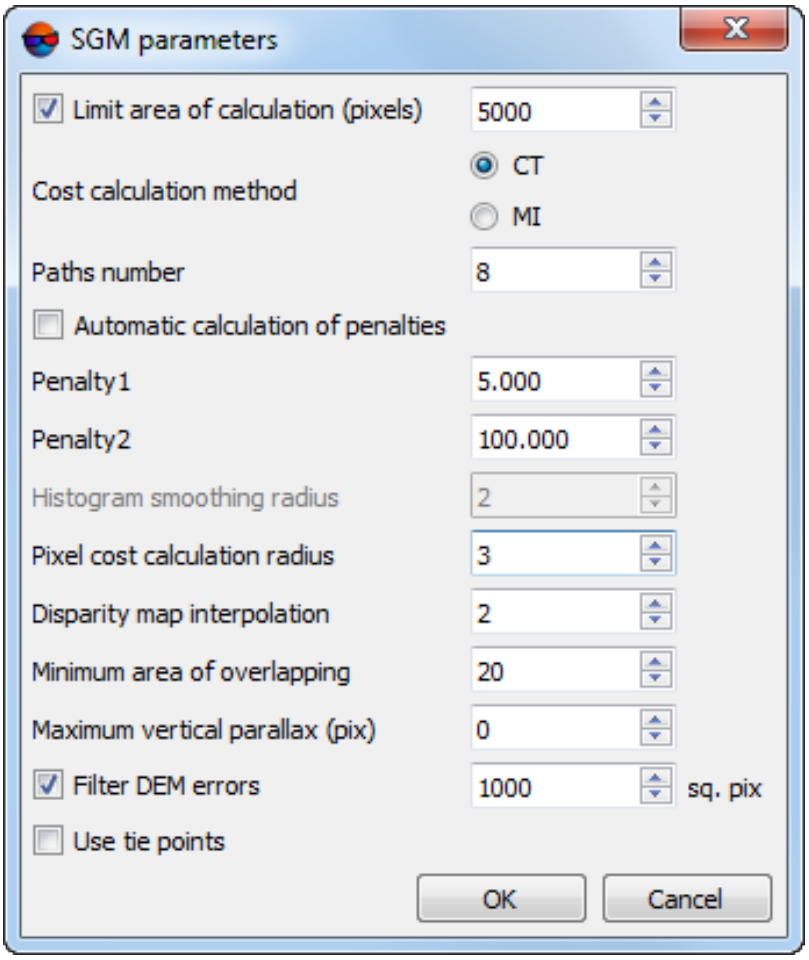


Fig. C.57. SGM parameters

Set the following parameters:

- **Limit area of calculation (pixels)** – specifies a size of parts that epipolar images will be splitted to. The parameter's value affects processing performance;
- **Cost calculation method** – **CT** (by default) or **MI** – this method is less sensitive to non-linear values of images brightness, but more demanding in performance (duration of DEM generation becomes 2-3 times longer);
- **Paths number** – number of calculation directions. It affects DEM accuracy and generation time. Recommended values: 8, 16, 32;
- **Penalty 1** – penalty for parallax change by 1 pixel. It is recommended to increase this value when using extensively noisy images. The value may be reduced to improve the accuracy of the DEM when using qualitative data;

The system allows to perform the **automatic calculation on penalties**. To set the **penalty 1** and the **penalty 2** manually clear this checkbox.

- **Penalty 2** – penalty for parallax change by more than 1 pixel. It is recommended to increase this value when using extensively noisy images. The value may be reduced to improve the accuracy of the DEM when using qualitative data;
- **Histogram smoothing radius** – parameters used just for calculation by **MI** method (see above). Defines smoothing level of epipolar images histogram;
- **Pixel cost calculation radius** – half-size of rectangular correlation mask in pixels by X and Y axes;
- **Disparity map interpolation** – interpolation distance when auto-filling missed results (null cells) by adjacent pixels;
- **Minimum area of overlapping** – minimal overlap area of epipolar image parts in pixels. Recommended value: not less than 20;
- **Maximum vertical parallax**, in pixels;
- **Filter DEM errors** – allows us to delete isolated DEM areas (having sizes lesser than set ones) during individual processing of each stereopair;

  Isolated DEM areas – the cells with values beyond the boundaries of the main DEM or among null cells.

- **Use tie points** – allows to improve DEM quality through the creation of temporary triangulation points before DEM construction to refine the elevation ranges of each stereopair involved in the construction.

  It is recommended to **use tie points** in case when the stages of tie and ground control points' coordinate measuring were excluded from the project preprocessing, and the preliminary adjustment necessary for dense DEM generation using the SGM method was carried out according to imported exterior orientation parameters.

  The creation of temporary triangulation point before DEM creation also can improve the quality of output data in case of using data of initial imaging with large convergence angles in stereopairs. Note, that preliminary creation of temporary triangulation points is quite time-consuming.

9. To increase the system performance through graphic processing unit resources, set the **Use GPU** checkbox and select the desired device from the drop-down list;

   The intensity of GPU usage can be monitored using various free software. *Windows 10* users can monitor GPU activity through the **Windows Task Manager** (**Performance** Tab).

   It is recommended to use modern graphics adapters with at least 4-6 GB of memory, especially in the case of parallel execution of several tasks in distributed processing mode.

   It should be considered that the performance of a video adapter as an extra device for computations is not directly dependent on its memory amount but is influenced by the al-

gorithms used and features of the system architecture.If there is a choice, first of all, *NVidia* video adapters are preferable.

During distributed processing, video adapters are to be selected in the **Monitor for Distributed Processing** window, individually for each computer used as a distributed processing client (see “Computers” in the “General information” User Manual. However, video adapters’ resources will be used for distributed computations only if the **Use GPU** checkbox is set in the current window.

If the computer used to configure the operation options (*server*) is also used as one of the *clients*, the selection of the video adapter used by it is also carried out through the **Monitor for Distributed Processing**, regardless of which device was selected in the drop-down list in the current window (this choice is taken into account only in the case of standard data processing mode).

10. [optional] To remove processing data after the calculation is completed set the **Delete intermediate data** checkbox on. Click the **(estimate the size)** button to get information about disk space occupied by intermediate data.

It is not recommended to remove intermediate data up to final creation of DEM. After removing of intermediate data it is impossible to start separate steps (filtering or smoothing) without DEM re-calculation.

11. In the **output data** section click the ... button that corresponds to the **Working folder** field and choose an *empty* target folder in active profile resources to save the output DEM.

If the selected folder already contains some data, before the execution of an operation the system will offer to decide between emptying the folder or using the existing data for DEM creation.

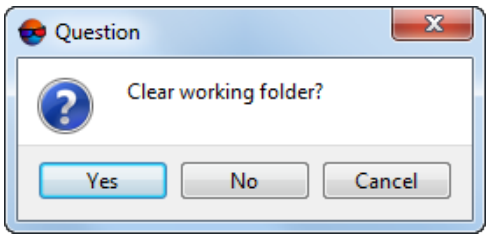


Fig. C.58. Question

12. Specify **DEM cell size** in meters to change size of output DEM cell.

The value of the **DEM cell size** parameter should not be less than average pixel size (GSD). When using cell size with smaller size the time of DEM creation and a size of output file are increasing, however, the processing accuracy is not improved at that.

Click the **From project** button to setup DEM cell size that equals to average GSD value in the project.

The final **DEM cell size** may change depending on detailed **output DEM parameters** settings (see below).

13. [optional] to **create** DEM, set the appropriate checkbox it the **DEM** section, and specify the following settings:

    - [optional] To set the additional **output DEM parameters** click the **Parameters** button in the appropriate section. The **Output DEM parameters** window opens:

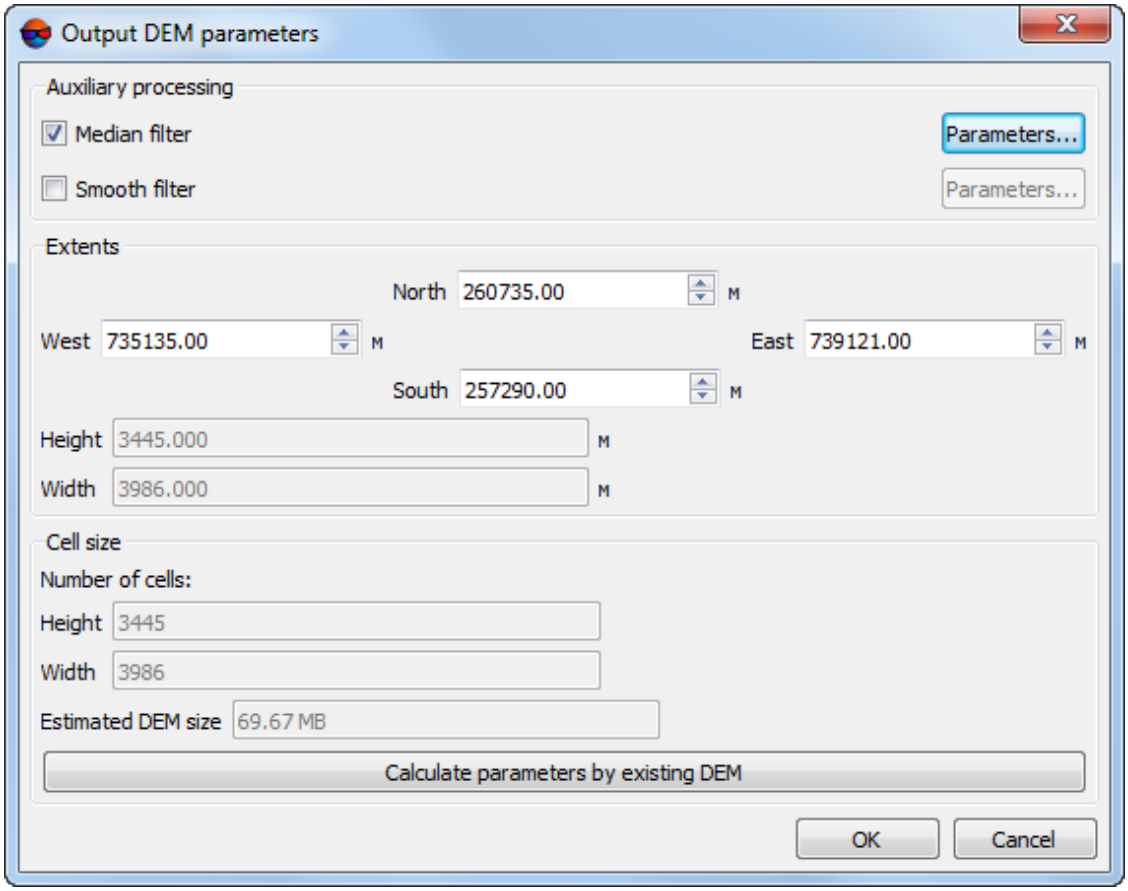


Fig. C.59. Output DEM parameters

    ○ [optional] In the **Auxiliary processing** section specify the following filtering parameters:

        ■ set the **Median filter** checkbox to use median filtering during DEM creation and specify **parameters** of filter usage;

        ■ set the **Smooth filter** checkbox to use smooth filtering during DEM creation and specify **parameters** of filter usage.

    ○ Specify DEM **boundaries** in the **North**, **West**, **East**, **South**. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters.

The **Height** and **Width** fields in the lower part of the window display the **number of cells** of the DEM (if it would be created with current **DEM cell size** and **boundaries** settings). There is also displayed the **Estimated DEM size** in megabytes.

In order to **Calculate parameters by existing DEM**, click the appropriate button and select existing DEM file in active profile resources. Click OK to apply these parameter and close the **Output DEM parameters** window.

- Specify the **Minimum number of counts in overlapped area** parameter value to setup reliability of DEM.

  This parameter determines the minimum number of overlapped stereopairs required for calculating the resulting DEM cell value.

  The cell of the final DEM will be recorded if the number of overlapped stereopairs used for its calculation is equal to the preset value (or exceeds it).

  Correct choice of this parameter value depends on minimum stereopair overlap in a block and on using interstrip stereopairs during DEM calculation (see above).

  Recommended default value is 2. In case of a small overlap of stereopairs, it is recommended to lower the value of the **Minimum number of counts in overlapped area** to 1.

  To *estimate* stereopair overlap area in the project, choose **Block › Create overlap map...** (see “Creating overlap map” of the “Creating project” User Manual).

  Function **Block › Create overlap map...** allows only to *estimate* stereopair overlap in the project, since map creation is carried out for *images* but not for *stereopairs*.

- Set the **Coefficient from RMS to Max error**, the parameter influencing the cell value filtering when merging the final DEM from fragments created using separate stereopairs.

  The **Coefficient from RMS to Max error** participates in the calculation of merged DEM fragment cell value tolerance. Decreasing this parameter’s value allows to build a DEM with higher accuracy but it may cause the increased number of null cells. Increasing this parameter’s value leads to the smoothing of DEM tile cell values during their merging.

  Computation of a resulting DEM in overlap areas of several fragments calculated by separate stereopairs is as follows:

  - Calculating the RMS of deviation from the mean elevation at the given cell;
  - Calculating a threshold value for the above RMS that is a product of **Coefficient from RMS to Max error** and **DEM cell size** parameter values.
  - If RMS falls within the above threshold, the mean value of the cell elevation calculated under these conditions is used as the resulting; otherwise, step-by-step filtering of fragments involved in given cell processing takes place until an acceptable result is achieved.

- Set the output DEM file**name**.

14. [optional] To **create** the **True Ortho**, limited by DEM boundaries, set the appropriate checkbox and specify the output **true ortho** file**name**. The **True Ortho** will be created in the **work folder**, in the PHOTOMOD MegaTIFF (`*.prf`) format;

To create the **true ortho** set the **create** DEM checkbox beforehand (see above).

Creating the true ortho will notably (approximately 2-fold) increase the time required for DEM creation.

PHOTOMOD GeoMosaic program allows to save later the obtained true ortho in any of available formats (see the "Orthophotomaps creation" User Manual).

For more detailed setting of true ortho creation options, click the **Parameters…** button. The **True ortho parameters** window opens:

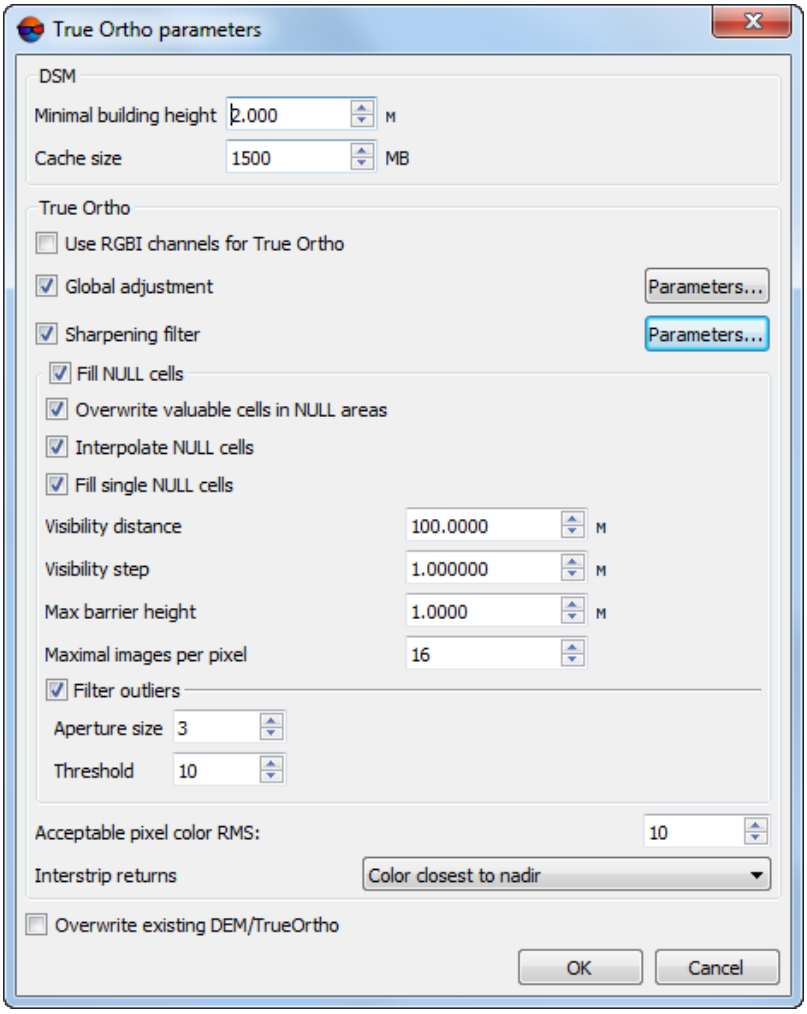


Fig. C.60. The Fill NULL cells window

- In **DEM** section, set **Minimum building height** in meters to exclude buildings from interpolation of DEM null cells (see the figure below);
- In **DEM** section, set **Cache size** when filling option in megabytes – maximum cache size allocated for a single task when creating true ortho;
- To **Use RGBI channels for true ortho** creation, set the appropriate checkbox;

A True Ortho creation using four channels is possible only when the images used as source data have the 4th channel, e.g. infrared.

- In the **True Ortho** section, set the **Global brightness adjustment** checkbox to apply global brightness adjustment over image overlapping areas when creating an orthophotomap. For more detailed setting of brightness adjustment options, click the **Parameters…** button. The **Parameters** window opens:

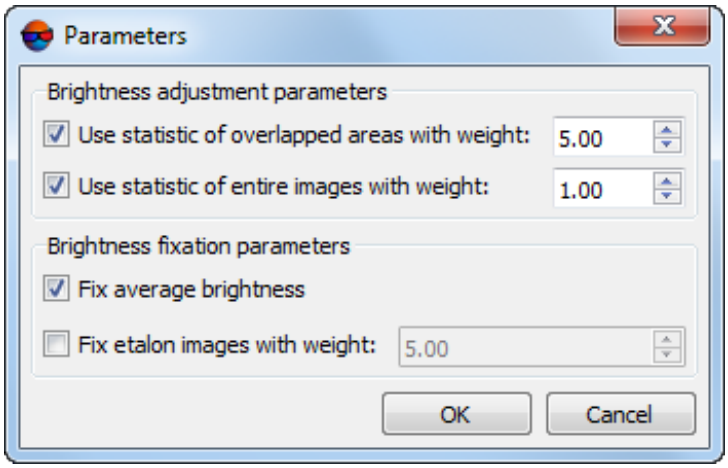


Fig. C.61. Brightness adjustment parameters

The opening window allows us to change the following settings of global brightness adjustment:

- **Use statistic of overlapped areas with weight** – allows to use the weight of the images' overlapping areas only;
- **Use statistic of entire images with weight** – allows to use the weight of all the pictures in the block;

If the images' brightness in a block significantly differ, it is recommended to set a small value.

- **Fix average brightness** – allows to save average brightness of images in block after global brightness adjustment;

- **Fix etalon images with weight** – allows to save (with preassigned weight) brightness of reference (etalon) images;

See the detailed description of brightness adjustment processes when creating an orthophotomap in the "Orthophotomaps creation" User Manual.

- [optional] to apply the **sharpening filter** set the appropriate checkbox and click the ... button to open the **Filter parameters** window:

The sharpen filter allows to highlight and intensify differences between image's individual details (image sharpness). See the "Radiometric correction" chapter of the "General information" User Manual.

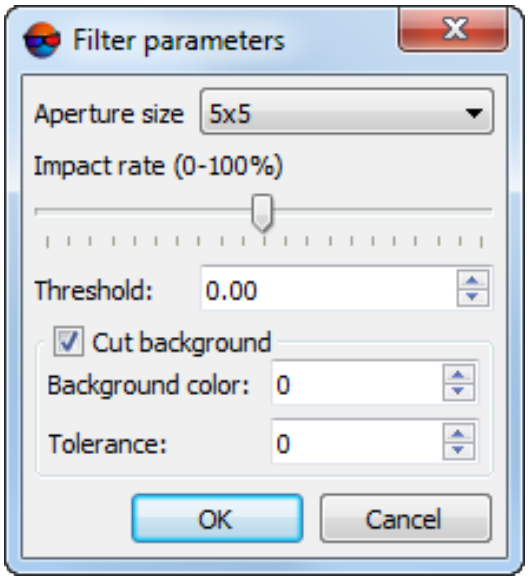


Fig. C.62. Filter parameters

Set the following parameters:

- In the **Aperture size** list define the matrix size from 3x3 pixels to 21x21 pixels.
- Move slider to define **Filtering level** in percent.
- Set the **Threshold**. Click OK to close the window.

Each single image element is filtered only if the brightness difference between adjacent elements exceed the given filtering **Threshold**. This helps to avoid errors looking as spots in the areas of an image with originally uniform hues.

- [optional] clear the **Fill NULL cells** checkbox in the **True Ortho** section in order not to fill null cells when creating a DEM (what can lead to creation of unfilled areas when creating an orthophotomap) or specify the following parameters:

The name of unfiltered output DEM with unfilled null cells has to look like the following: `<DEM name>_tmp.x-dem` (automatically set).

The name of output True Ortho with non-interpolated null areas has to look like the following: `<True Ortho name>_tmp.prf` (automatically set).

- **Overwrite valuable cells in NULL areas** – allows to exclude *isolated DEM areas* from interpolation of DEM null cells (i.e. interpolate these areas as a NULL cells);

  Isolated DEM areas – the cells with values beyond the boundaries of the main DEM or among null cells.

- **Interpolate not filled NULL cells** – allows to interpolate unfilled DEM/orthophotomap areas;

- **Fill single NULL cells** – allows to interpolate areas without data having one pixel size in a DEM or orthophotomap;

- **Visibility distance** – the distance within which the visibility of points located in unfilled areas of an orthophotomap is checked in the project's images;

- **Visibility step** – an increment with which the visibility of points located in unfilled areas of an orthophotomap is checked in the project's images;

- **Maximal barrier height** – an approximate maximal height of the objects towering above the Earth surface and hiding some terrain areas (see the illustration below);

- **Maximum images per pixel** – maximum number of images of the project where the visibility of points located in unfilled areas of an orthophotomap is checked;

  The system calculates default **Visibility distance**, **Visibility step** and **Maximum images per pixel** based on the **DEM cell size** specified by the user (see above).

  If, after closing the **Fill NULL cells** window, the user changes the **DEM cell size** in the **SGM parameters** window, **Visibility distance**, **Visibility step** and **Maximum images per pixel** manually set by the user will be automatically recalculated.

  When setting parameters for null cell filling, the survey geometry (off-nadir angles) is to be taken into account, as well as the geometry of the terrain itself (heights of objects located on the terrain). If the results of cell filling are found to be unsatisfactory, increase **visibility distance** and **visibility step** (given that this will increase the time spent on the operation).

- **Filter outliers** – allows to perform orthophotomap median filtering with preset **Threshold**. The statistics is collected in the neighborhood whose area is de-

termined by the **Aperture size**. This filter is used for removing single pixels with unnatural brightness;

- Set the **Acceptable pixel color RMS**, the parameter influencing the filtering of pixels of various images when creating a true ortho.

  Decreased values of this parameter lead to exacting tolerance for pixel colors of various images, and, as a consequence, may cause the decrease of the number of images used for the calculation of each true ortho pixel.

- Define the output **True Ortho** pixel **color selection method**:

  The color of each **True Ortho** pixel is calculated over several images involved in the processing of a particular pixel. By setting an **acceptable pixel color RMS**, the user can pre-exclude from the processing those images, the color of a particular pixel of which differs significantly from the corresponding pixels of the other images.

  - Use the **color closest to nadir** – only one image with the smallest deviation from the nadir angle, relative to a given pixel is involved in determining the resulting color of a pixel;

    When using this technique, well-marked differences between the colors of adjacent pixels may be observed in the output **True Ortho**, even if **global brightness adjustment** is used (see above).

  - Use the pixel color of one of the images that is **closest to average color** calculated for this pixel (according to all images involved in processing);

  - Use **weighted color selection**;

  - **Set average color** (average the data of all images involved in the pixel calculation)

- [optional] set the **Overwrite existing DTM/TrueOrtho** checkbox in order not to use intermediate data from the working folder (if DEM and orthophotomap has already been created) but overwrite them.

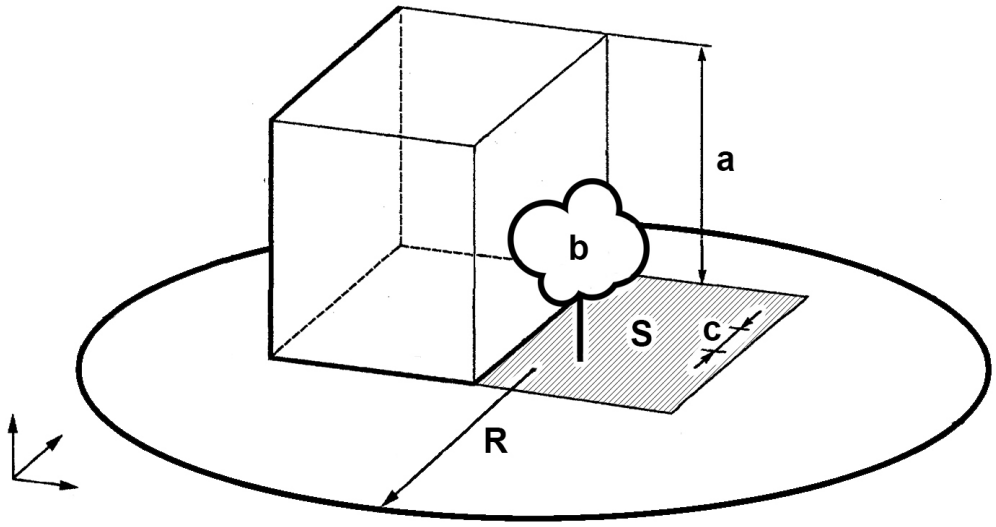


Fig. C.63. An example of unfilled area on true ortho, where **a** is the building height (see **Maximal barrier height** parameter), **b** is the "isolated DEM area" among DEM NULL cells (see **Overwrite valuable cells in NULL areas** checkbox), **c** is the **Visibility step**, **R** is the **Visibility distance**, **S** is the unfilled DEM/orthophotomap area (due to the fact that the image is the central projection of terrain and objects towering above the Earth surface hide some terrain areas, thus they are not seen on the image). Also, the quality of an orthophotomap is affected by the zones hidden by the object's shadow, and brightness adjustment is required.

15. [optional] To **create** an output **point cloud (LAS)** set the appropriate checkbox (see the "LIDAR Data processing" User Manual). For more detailed setting of point cloud creation parameters, click **Parameters...** button. The **LAS creation parameters** window opens:

If the **Create point cloud (LAS)** checkbox is set, it is recommended to use the distributed processing mode for dense DTM creation using the SGM method.

Points cloud will be saved in the separate LAS directory in the folder chosen for output DEM saving in the active profile's resources.

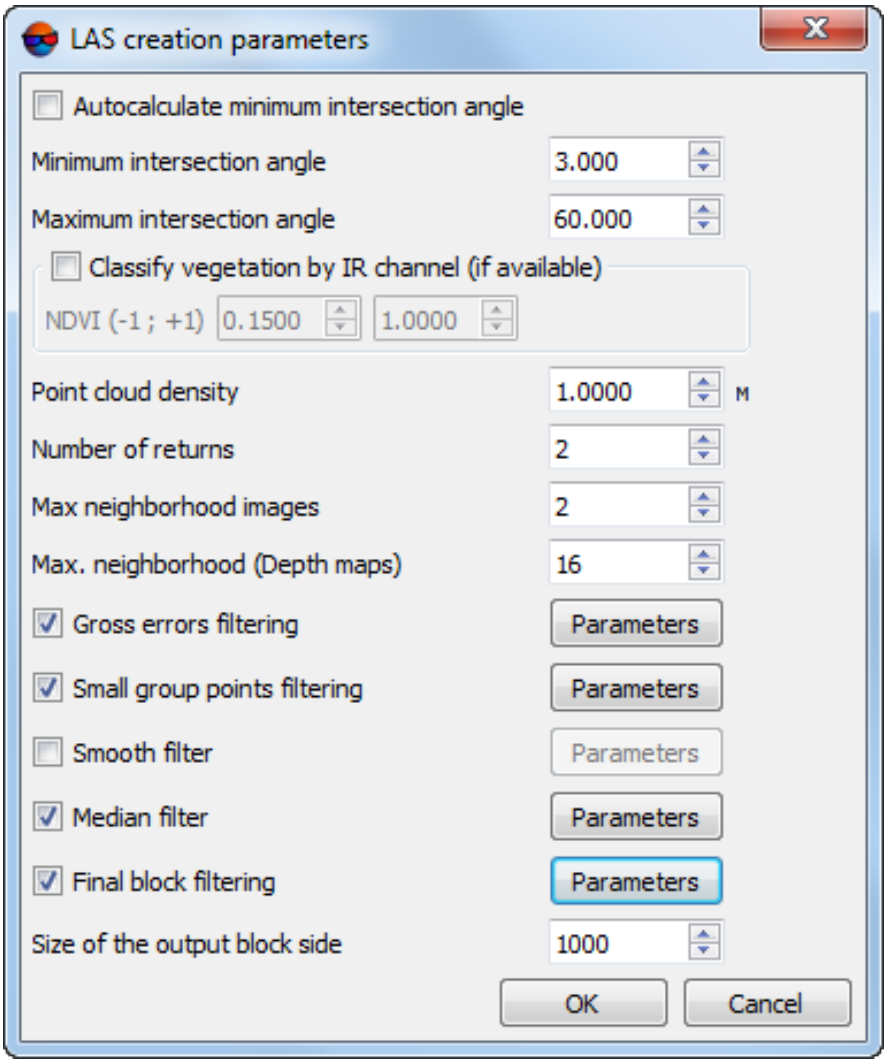


Fig. C.64. The Create LAS parameters window

- Set the **Maximum intersection angle** and **mininum intersection angle** in degrees to filter blunders by Z on images with intersection angle of small or null value;

Before configuring these parameters, it is recommended to assess the overlapping of the project images as well as the intersection angles of the stereopairs used in data processing.

To estimate the project images overlaps, choose **Block › Create overlap map...** (see “Creating overlap map” of the “Creating project” User Manual).

The system provides the different methods to estimate the intersection angles in the stereopairs of the project (see the “The list of stereopairs for a selected image” chapter in “Creating project” User Manual and the “Creation of “stereo quality” map” chapter in “Vectorization” User Manual).

To set the **minimum intersection angle** among all stereopairs used in data processing as a limiting value, set the **autocalculate minimum intersection angle** checkbox.

In most cases, especially when processing data with a large number of overlaps and multiple small intersection angles, leave this checkbox set

Increasing the **mininum intersection angle** manually, user can achieve more LAS accuracy due to tighter point filtering.

- To **classify vegetation by IR channel (if available)** set am appropriate checkbox (see the “Vegetation classification” chapter of the “LIDAR Data processing” User Manual);
- Set **Point cloud density** in meters;

By default, **Point cloud resolution** is twice as large as the **DEM cell size** (see above).

It is not recommended to set **Point cloud resolution** lesser than the **DEM cell size**.

- Specify the **Number of returns** – the minimum number of stereopairs required for point calculation;
  - Specify the **Max neighborhood images** number – the maximum number of images used for stereopairs creation.
- To perform the preliminary **gross errors filtering** (for the LAS fragments calculated for the single images) set an appropriate checkbox and click the appropriate button to set filtering **parameters** (see the “LAS statistical outlier removal” chapter of the “LIDAR Data processing” User manual);
- To perform the **small group points filtering** set an appropriate checkbox and click the appropriate button to set filtering **parameters**:

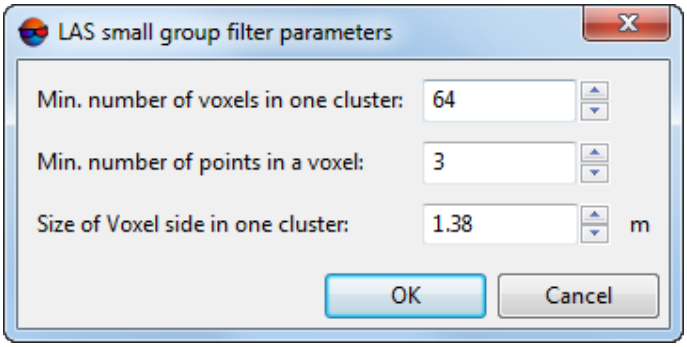


Fig. C.65. The “LAS small group filter parameters”

  - **Min number of voxels in one cluster** – if the number is less, the cluster will be deleted;
  - **Min number of points in a voxel**;

- **Size of Voxel side in one cluster** – the side size of the single voxel in project units.

The **small group points filtering** allows to delete point groups out of the main cloud which are located at some distance and most likely are noisy.

A volumetric pixel (*voxel*) is the three-dimensional equivalent of a pixel and the tiniest distinguishable element of a 3D object. Volume pixels are used like building blocks to form a larger 3D object.

In this case, a *voxel* is a small fragment of the point cloud, within which the calculations are performed. Not to be confused with the LAS point cloud tile.

The *cluster* is the group of *voxels*.

- To use the **smooth filter** for the output points cloud set an appropriate checkbox and click the appropriate button to set filtering **parameters** (see the “LAS fusion” chapter of the “LIDAR Data processing” User manual);

- To use the **median filter** for the output points cloud set an appropriate checkbox and click the appropriate button to set filtering **parameters** (see the “LAS fusion” chapter of the “LIDAR Data processing” User manual);

- To perform the **final block filtering** (statistical outlier removal for the output points cloud) set an appropriate checkbox and click the appropriate button to set filtering **parameters** (see the “LAS statistical outlier removal” chapter of the “LIDAR Data processing” User manual);

- Specify the **size of the output block side** to set the output LAS tile size. The default **size of the output block side** is 1000 points cloud GSD (see the **Point cloud density** parameter above).

16. Click OK to start dense DEM generation operation in normal mode. Output DEM is opened in a new layer after the creation is finished. Output true ortho is also opened in a new layer if the corresponding checkbox was turned on.

    In order to generate DEM using distributed processing, perform the following actions:

While building DSM by SGM method in distributed processing mode, stereopairs are processed separately in different tasks (see the **Searching area** section). The last task is responsible for merging data acquired.

1. Start the server/client of distributed processing mode (see the “Distributed processing” chapter in the “General information” User Guide).

2. Open the **Monitor for distributed processing** and specify **Max MT tasks** – the maximum quantity of simultaneously running tasks on *Client* in *MultiThreading* mode;

MultiThreading – in computer architecture, multithreading is the ability of a central processing unit (CPU) (or a single core in a multi-core processor) to provide multiple threads of execution concurrently, supported by the operating system.

This approach differs from multiprocessing. In a multithreaded application, the threads share the resources of a single or multiple cores, which include the computing units, the CPU caches, and the translation lookaside buffer (TLB).

Where multiprocessing systems include multiple complete processing units in one or more cores, multithreading aims to increase utilization of a single core by using thread-level parallelism, as well as instruction-level parallelism. As the two techniques are complementary, they are sometimes combined in systems with multiple multithreading CPUs and with CPUs with multiple multithreading cores.

When setting this option, it is needed to take into account the *Client's* specifications i.e. **RAM** and the number of processor cores. The recommended **Max MT tasks** value for a workstation having a multicore processor and 8 GB **RAM** is 1. The recommended **Max MT tasks** value for a workstation with 16 GB **RAM** is 2, etc.

3. Click the **Distributed processing...** button.

Depending on the size of creation area the operation can take a long time.

To provide a system performance while building dense DSM by SGM method it is recommended to use Distributed processing mode.

C.7.4.3. Slope based filter

The system allows to filter objects based on slope angle.

While using a filter, DEM cells covering buildings, vegetation, man-made objects and accident spikes are searched for and removed. As a result of filtering DEM contains heights of "true relief" only.

1. Make active a DEM layer;
2. Select **Terrain › DEM › Filter › Slope based filter…** or click **Slope based filter** button on the **Compute DEM** tab of the main toolbar. The **Slope based filter** window opens;

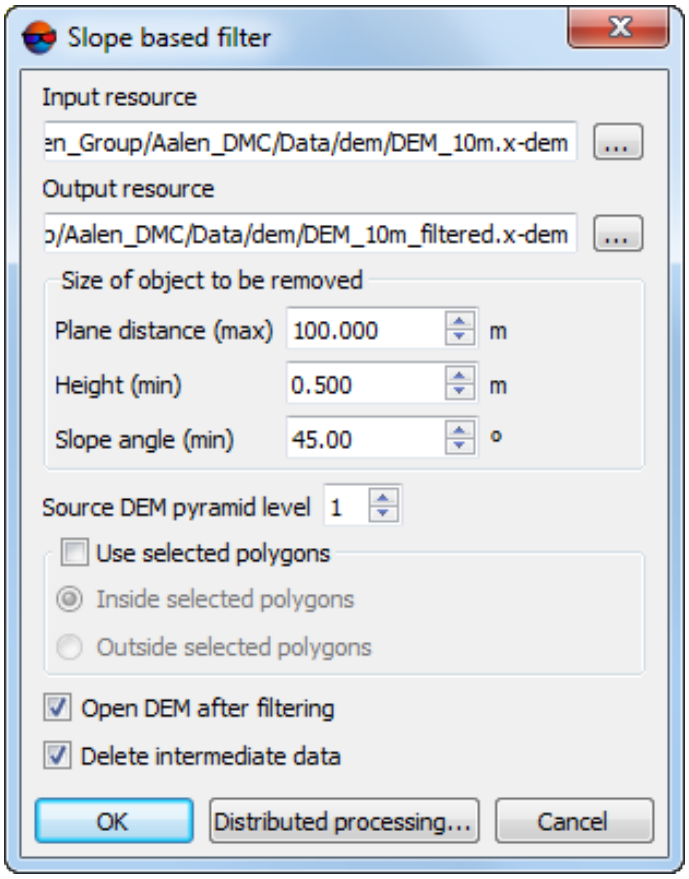


Fig. C.66. Parameters of slope based filter

3. In **Input DEM** section click ... button and select DEM to filter in resources of the active profile;

4. In **Output DEM** section click ... button and specify name and target folder to keep output DEM in resources of the active profile;

5. In **Size of object to be removed** section specify the typical object size in XY-plane and typical object size along Z axis in following fields in meters:

   - **Plane distance** (max);
   - **Height** (min).
   - **Slope angle** (min) in degrees.

   A **slope angle** is one of the parameters used to calculate a threshold for filtering. For example, for a plumb-wall building, the **slope angle** is ~ 90 degree (note, that due to the peculiarities of DEM construction, this angle will never be exactly 90 degrees).

   The **slope angle** is selected individually in each case for the distinctive buildings and objects that need to be removed. Large values of slope angles (about 80 degrees) may

lead to the fact that those buildings of specific shapes (e.g. hangars) may not be removed during filtering. Decreasing the slope angle (e.g. to 45 degrees) may lead to removing some accidents of terrain (e.g. elevations having similar shapes and sizes) along with abovementioned buildings.

6. To perform initial filtration of a thinned original DEM, set the **Intermediate DEM pyramid level** determining the degree of thinning.

   Increasing the degree of thinning makes it possible to significantly improve the system processing performance, lowering the quality of filtration. Recommended values of **Intermediate DEM pyramid level** are 1, 2, or 3.

   To set zero value is strongly discouraged.

7. [optional] Set **Use selected polygons** checkbox to filter DEM in the borders of selected polygons only. Select where filtration will be performed:

   - **Inside selected polygons**;
   - **Outside selected polygons**.

   To **Use selected polygons**, before opening **Slope based filter** window perform the following:

   - Create vector layer (see the "Vectorization" User Manual);
   - Create one or several polygons which limit selected fragments of DEM or load vector layer with polygons to use them as boundaries for filtering;
   - Select no less than one polygon which is used as a boundary for filtering.

8. [optional] Set **Open Dem after filtering** checkbox to open output DEM in a new layer immediately after filtration completion.

9. [optional] To remove processing data after the calculation is completed set the **Delete intermediate data** checkbox on.

10. Click OK. A process of filtering DEM is started. Output DEM is opened in a new layer if the corresponding checkbox was turned on.

    To use distributed processing for DEM slope based filtering, do the following:

    1. Change settings and run the distributed processing server/client (see the "Distributed processing" chapter in the "General information about system" User Manual).
    2. Click the **Distributed processing** button.

C.7.4.4. Filling null cells using smooth interpolation

The system provides possibility to restore null cells of DEM using the smooth interpolation method. When applying the method of smooth interpolation the system evaluates the factor of input DEM thinning to obtain auxiliary DEM. An auxiliary DEM is used to calculate elevation values of null cells, and DEM cells with filled elevations are not re-calculated. Borders and cell size of output DEM are fully correspond to the boundary and the cell size of the input DEM.

This method is recommended for a sufficiently dense DEM.

To fill DEM null cells by smooth interpolation method perform the following methods:

1. Select **Terrain › DEM › Fill null cells › Smooth interpolation...** or clock **Slope based filter** button on the **Compute DEM** tab of the main toolbar. The **Parameters of filling null cells in DEM** window opens.

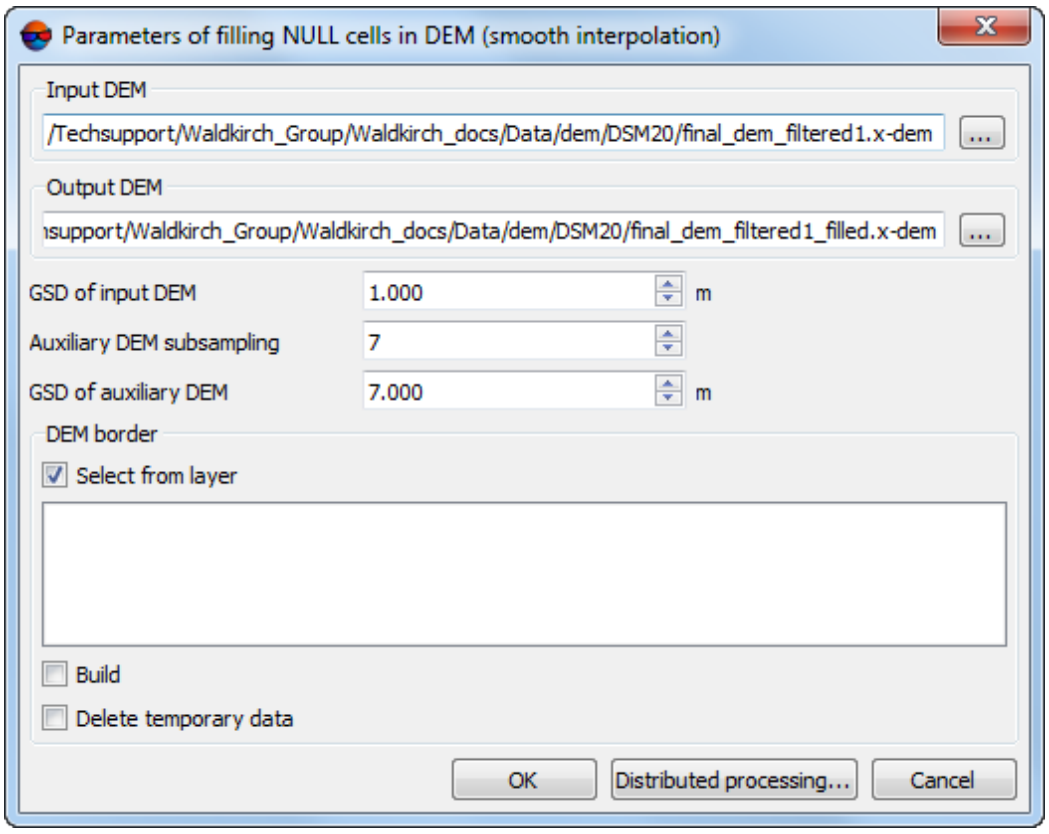


Fig. C.67. Parameters of filling null cells using smooth interpolation

The window displays the **ground sample distance (GSD)** of input DEM in meters.

2. In the **Input DEM** section click the ... button to select input DEM in active profile resources.

The DEM loaded to the project is selected by default.

3. [optional] To define name and target folder of output DEM in active profile resources, click the ... button.

By default the system suggests the *<input dem name>_filled.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. Input the **Auxiliary DEM subsampling**. After that the **GSD of auxiliary DEM** is calculated automatically.

The **Auxiliary DEM subsampling** must be an odd number.

5. [optional] In the **DEM border** section, set the **Select from layer** checkbox to set border of the output DEM using a vector polygon.

It is required to create in advance a vector layer with *one* polygon which sets borders of the output DEM.

From the opened list of downloaded vector layers, select *one* vector layer that contains a polygon limiting a DEM;

6. [optional] In the **DEM border** section, set the **Build** checkbox to create borders of the output DEM automatically;

7. [optional] To remove processing data after the calculation is completed set the **Delete intermediate data** checkbox on;

8. Click OK. The system starts calculation of null cells elevation using smooth interpolation method. After that the system creates new DEM layer in the *Manager*.

To use distributed computing for DEM rebuilding, do the following:

1. Change settings and run the distributed processing server/client (see the “Distributed processing” chapter in the “General information about system” User Manual).

2. Click the **Distributed processing** button.

C.7.5. Build contours

Refer to the “Contour lines” chapter of the “DTM Generation” User Manual for the detailed information about contours generation.

C.7.5.1. Generation contour lines by DEM

The system provides possibility to create contour lines by digital elevation model (DEM).

In order to build contours using DEM perform the following actions:

1. Load a DEM layer.
2. [optional] Create and select one or multiple vector polygons, if it is needed to build a contours inside selected polygons;
3. Choose **Terrain › Contours › Build contours › From DEM...** or click **Build contours** button on the **Compute DEM** tab of the main toolbar. The **Contours creation parameters** window opens.

   The **DEM parameters** section displays information about elevation difference (**Min. elevation**, **Max. elevation**) in DEM.

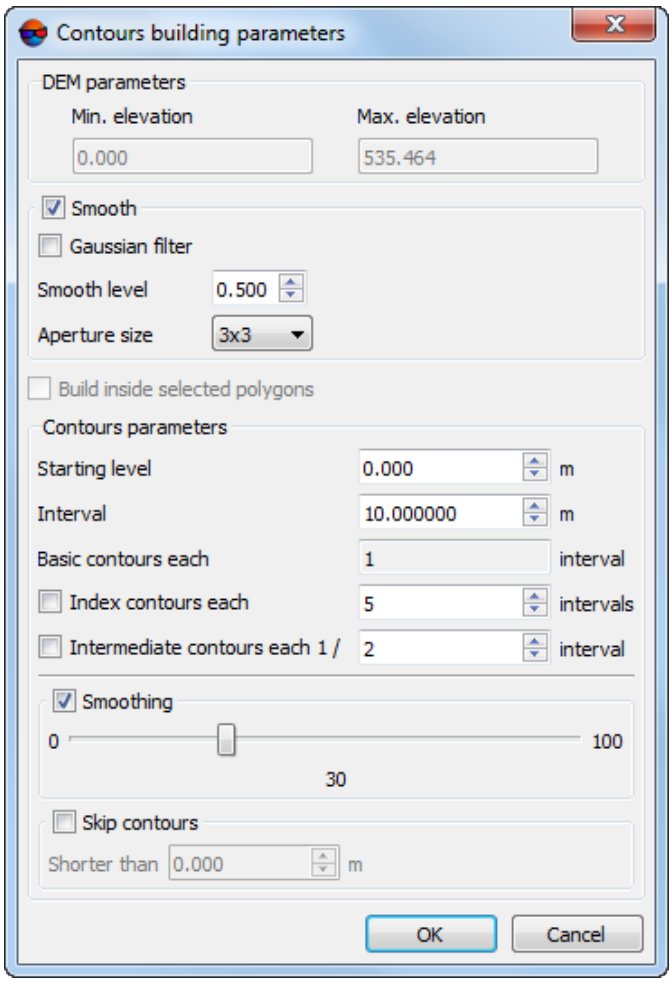


Fig. C.68. Parameters of contour lines generation by DEM

4. [optional] In order to perform the of DEM smoothing operation, prior to contours creation, set the **Smooth** checkbox on in the **DEM** section. Specify smoothing level in the **Smooth level** field (maximal smoothing is applied at 1 value).

   During smoothing the system uses information about adjacent nodes in each DEM node. In order to specify number of adjacent nodes of DEM, select the **Aperture size** in the list.

5. [optional] To apply Gauss smoothing algorithm set the **Smooth by Gauss** checkbox;

6. [optional] Set the **Build inside selected polygons** checkbox to create a contours within the selected polygons;

7. [optional] Input minimal elevation level (Zmin), which will be zero-level for contours creation, to the **Starting level** field.

   In order to obtain information about about elevation difference (Zmin, Zmax), select **Edit › Active layer › Layer information** or click the DEM layer by mouse right button in the *Manager* and select **Information**. The system displays the information window that contains information about the layer.

8. In the **Interval** field specify contours interval in meters. Interval of **Basic contours** equals to 1 contours **Interval** (displayed in the **Basic contours** field).

9. [optional] To create index contour lines the **Index contour lines** checkbox is on by default. Set the checkbox off to cancel index contour lines creation.

   The interval of **index contour lines** equals to five intervals of basic contours by default. The system allows to set arbitrary interval of **index contour lines** manually.

10. [optional] To create intermediate contour lines the **Intermediate contour lines** checkbox is on by default. Set the checkbox off to cancel intermediate contour lines creation.

    The interval of **intermediate contour lines** equals to half of basic contours interval by default. The system allows to set arbitrary interval of **intermediate contour lines** manually.

11. [optional] In order to create contours as smoothed curves, set the **Smoothing** checkbox on and specify a level of smoothing using the slider.

12. [optional] In order to cancel creation of short contours, set the **Skip contour lines shorter than** checkbox on and input minimal contour length in meters.

13. Click OK. After that the system starts to create contours in a new layer *Contours*.

C.8. Orthorectification

Orthoimage is a georeferenced image prepared from a perspective photograph or other remotely-sensed data in which displacement of objects due to sensor orientation and terrain relief have been removed.

The system provides possibility of creating orthoimages by source images block. This step allows to prepare images for further processing and creating mosaic in the*Geomosaic program*program.

During *orthorectification* the system corrects relief displacement, axial angle, camera distortion, etc. For images orthorectification it is necessary to specify output orthophoto cell size, select output image coordinate system and scale, specify output file format and georeference type.

Prior to create orthoimages it is necessary to adjust images block (see “Block adjustment” User Manual).

The **Orthorectification** toolbar is used for creating orthoimages.

In order to display the toolbar, choose the **Rasters › Orthorectification** (**Ctrl+Alt+M**) or click the  button on the main toolbar.

The following elements are displayed in the **Orthorectification** toolbar title:

- A toolbar name: **Orthorectification**;
- [optional] The “*” symbol meaning that current orthorectification settings have been changed, and hence the orthorectification project is to be saved or overwritten (, );

  If the project is not saved before orthorectification start, an appropriate info message is issued before the operation that offers to save this project (see below).

- [optional] Current orthorectification project name.

  The project name is in quotes, after the window name and the “*” symbol (which, in turn, is displayed if the project has unsaved changes)

Table C.1. The Orthorectification toolbar

Buttons	Function
	to create the new orthorectification project
	to open an orthorectification project from active profile resources
	to save an orthorectification project
	to save an orthorectification project with a new name
	to open the **Images list** window (see the ’Creating project’ User Manual)
	to set the orthorectification parameters

Buttons	Function
	to set the percent of trim image edges during orthorectification manually
	to specify the percent of trim image edges during orthorectification by setting the residual overlap
	to perform the accuracy control of orthorectification
	to start of orthoimages building and creation of output file for selected sheet
	to start orthoimages creation for specified active sheets considering user settings and parameters in distributed processing mode
	to start of orthoimages building in the MegaTIFF format using the distributed processing mode
	to create a separate layer with orthoimage for each of selected images
	to measure a building height (see the “Vectorization” User Manual)
	to show general information about project (number of images channels, byte per channel, number of images and output size of mosaic)
	to move to previous image in a block scheme. The first image of the first strip opens if the block scheme 2D-window is active
	to open selected in 2D-window of a block scheme image in a separate window
	to move to next image in a block scheme. The last image of the last strip opens if the block scheme 2D-window is active
	to close all separate raster *FastOrtho* layers

 Refer to the “Orthorectification” User Manual for the detailed information about orthorectification.

Appendix D. Input and output data

Input data

During the project creation, aerial triangulation and block adjustment the following source data is used:

- [optional] The *list of GCP coordinates* is necessary for block adjustment. The system provides opportunity to measure GCP coordinates manually and also to perform import of GCP list from a text file.
- [optional] The *list of projection centers* coordinates to process central projection images. The program also provides the ability to import on-board data (coordinates) about projection centers from CSV files.

 Block adjustment could be process only by projection centers without using ground control points. It is recommended to use GCP coordinates to increase adjustment accuracy.

- *Raster images*

For working with images in program , it is recommended to use MS-TIFF internal raster format, which is the TIFF format with overview pyramid (set of subsampled images copies) for higher image display performance.

The *Raster Converter* module is used to preliminary raster image processing. It is possible to convert image into internal format both manually, before creating project, and on the stage of adding images into project with saving converted images into active profile resources. See the *Raster Converter module* description in the "General information" User Manual.

The following image formats are available to process central projection images:

- JPEG (JPEG);
- Tag Image File Format (TIFF) TIFF и GeoTiff format, included tags for saving of georeference information;
- Windows Bitmap File (BMP);
- Advanced Systems Format (ASF).

- *Information about coordinate system and map projection* – during project creation it is necessary to define coordinate system of GC points. By default there is an International coordinate system database and map projection (see the details in the "GeoCalculator" User Manual);
- also it is possible to use the following exterior data:
  - import of triangulation points from PAT-B and X-POINTS;
  - import of ground control points from text files `*.txt` and `*.csv` (see above *The list of GCP coordinates*);
  - import of interior and exterior orientation data from metadata;
  - import of external orientation data from PAT-B and CSV-files (see above *The list of projection centers*);
  - GPS data;
  - import additional data from different formats.

Output data

The program allows to process UAS data and acquire all types of value added photogrammetric products: DEM, 2D and 3D-vectors, orthomosaics.

There are the main output products:

- *Digital Terrain Model (DTM)* – digital cartographic presentation of terrain surface both as regular grid of elevations (DEM) and as triangulated irregular network (TIN). They are used for solving applied research problems.

- *3D vector objects* – used for creating a topographical base for cartographic production or as source data for a mathematical representation of a scene in three-dimensional digital terrain modeling;

- *orthophoto production* – single raster images in the form of a single frame or a set of sheets in a selected map projection with marginalia. In the resulting image, geometric and photometric distortions are corrected, creating seamless, color-balanced orthophotos with uniform brightness as an output;

- *3D models of terrain* – can be used to solve applied research problems, as well as for creation of multimedia presentations and commercials.

There is opportunity to export all photogrammetric products to various formats.

Appendix E. The PHOTOMOD8UAS.VAR configuration folder

At the stage of system first configuration is automatically created the *PHOTOMODUAS8.VAR* folder. This folder is used to store configuration, temporary and other system files.

Only one *PHOTOMODUAS8.VAR* folder creates, even for several installed copies of the system.

It is not recommended to place the configuration folder in the net, because it leads to slowing of system's work.

PHOTOMODUAS8.VAR folder contains the following files and folders:

- *AutoSave* folder – is used to store autosaved data:

- *Config* folder – is used to store files of general parameters of all profiles;

It is possible to return to default parameters in case of changing configuration file. To do this, remove a configuration file in the *PHOTOMODUAS8.VAR* folder and restart the system. Default configuration file are copied from the *PHOTOMODUAS8.VAR\Config* system files folder.

Both global settings and settings for the local profile is loaded at startup of the system and saved when you exit. Local project settings is loaded when loading a project and saved when project closing. In case of working without projects, settings are stored in the *PHOTOMODUAS8.Var\Profiles\[profile_name]\VoidProjOptions.x-ini* file.

- *Logs* folder – is used to store log-files for all profiles;

- *Profiles* folder – is used to store all parameters for each profile separately. The list of local and network profiles is also stored in the folder. It is the same list as in the **Control Panel** module (see the “Control Panel. Profiles management” chapter of the “General information” User Manual). Configuration file with resource structure and path to local/network folder is also stored in folder of each profile;

New subfolder creates for each profile.

- *Tmp* folder – is used to store temporary files;
- *UserData* folder – is used to store data out of resources system;
- *policy.x-ini* file – contains general information about configuration parameters (active profile name, name and path to centralized management folder and so on).