

Digital Photogrammetric System

PHOTOMOD

Version 8.1

USER MANUAL

DTM Generation
(Linux)

Table of Contents

1. Purpose of the document	6
2. DTM Generation	6
2.1. The "Terrain" menu	6
3. Regular grid of nodes	7
3.1. Grid creation	7
3.2. Grid parameters	9
4. Points	11
4.1. The "Points" menu	11
4.2. Points creation	12
4.2.1. Points creation modes and methods	12
4.2.2. The "Pathway mode" toolbar	13
4.2.3. Points creation in pathway mode	13
4.2.4. Parameters of pathway mode	15
4.3. Points automatic calculating	17
4.3.1. Automatic calculating of points	17
4.3.2. Correlator configurations	23
4.3.3. Setup of correlator configuration	24
4.3.4. Additional correlator's parameters	27
4.3.5. Parameters of first approximation calculation	29
4.3.6. Points calculation in distributed processing mode	30
4.4. Points editing	32
4.5. Points filtering	33
4.5.1. Points thinning	34
4.5.2. Points filtering by Z-range	35
4.5.3. Median points filtering by Z	35
4.5.4. Filter of adjacent point objects	36
4.5.5. Buildings and vegetation filter	37
4.5.6. Surface objects filter	45
4.5.7. Filter by image properties	46
5. Triangulation irregular network (TIN)	50
5.1. The "TIN" menu	50
5.2. General information	51
5.2.1. TIN layer display settings	52
5.3. TIN creation	53
5.3.1. Creation pre-regions for a block	53
5.3.2. Loading base layers for TIN creation	56
5.3.3. TIN borders creation	59
5.3.4. Building TIN	61
5.4. TIN display	62
5.5. TIN loading	64
5.6. TIN saving	65
5.7. TIN restoring	66
5.7.1. TIN restoring from contour lines	66
5.8. TIN info	69
5.8.1. Brief TIN info	69
5.8.2. General TIN info	69
5.8.3. TIN area info	71
5.9. TIN re-building	72
5.10. Insert objects into TIN	73
5.11. Accuracy control of TIN creation	74
5.11.1. Check against adjustment points	74
5.11.2. Check TIN against vector objects	77
5.11.3. Verifying of TIN topology	79

5.12. Export TIN	80
5.12.1. Export TIN to DXF	80
5.12.2. Export TIN to CSV	81
5.13. Import TIN	84
5.13.1. Import from DXF	84
5.14. TIN filtering	87
5.14.1. TIN simplifying	87
5.14.2. Peak filter	88
5.14.3. TIN filtering by Z-range	90
5.14.4. Smooth TIN interpolation	91
5.15. Converting TIN into a vector layer	93
5.16. Creation of textured TIN 3D surface	93
5.16.1. Transformation of 3D-TIN coordinate system	116
5.16.2. Batch 3D-TIN creation	119
6. Contour lines	121
6.1. The “Contours” menu	121
6.2. Contour lines generation	122
6.2.1. Generation contour lines by TIN	122
6.2.2. Generation contour lines by DEM	126
6.2.3. Contours creation in batch mode	128
6.3. Contour lines editing	130
6.4. Operations with contour lines	131
6.4.1. Saving contour lines	131
6.4.2. Export of contours sheets	131
6.4.3. Contours merging	136
6.4.4. Check of contours merging quality	138
6.4.5. Contours connecting	139
6.4.6. Precision of objects coordinates	141
6.5. Accuracy control of contours creation	142
6.5.1. Control of contours crossing	142
6.5.2. Check contours by points	144
6.5.3. Checking elevations of contour vertices	146
7. Digital Elevation Model	149
7.1. The “DEM” menu	149
7.2. General information	151
7.2.1. DEM layer display settings	152
7.3. DEM creation	153
7.3.1. DEM creation by TIN	153
7.3.2. DEM creation using regular points	155
7.3.3. DEM creation using points	156
7.3.4. DEM creation using CSV-file	157
7.3.5. Dense DEM generation using SGM method	159
7.3.6. DEM creation in batch mode	179
7.3.7. Building DEM from an external source	182
7.4. DEM filtering	186
7.4.1. Buildings and vegetation filter	186
7.4.2. Slope based filter	191
7.4.3. Median filter	193
7.4.4. Smooth filter	194
7.4.5. Bilateral filter	195
7.4.6. Filter by image properties	197
7.4.7. Selective filtering of objects	201
7.5. DEM classification	204
7.5.1. DEM cells classification	206

7.5.2. Extraction of classes from DEM	208
7.6. Transformation of DEM coordinates	210
7.6.1. DEM transpose	210
7.6.2. Transformation of DEM coordinate system	210
7.6.3. Batch transformation of DEMs coordinate system	214
7.6.4. Rebuilding DEM considering last adjustment	217
7.7. Null cells in DEM	219
7.7.1. General information	219
7.7.2. Filling null cells using linear interpolation	219
7.7.3. Filling null cells using smooth interpolation	222
7.7.4. Filling null cells by adjacent value	224
7.7.5. Filling null cells by constant value	227
7.7.6. Filling null cells by minimum values	228
7.7.7. Converting cells to null	231
7.8. DEM editing	232
7.8.1. DEM merging	232
7.8.2. Change DEM parts elevation	237
7.8.3. DEM shift	239
7.8.4. Split DEM into sheets	242
7.8.5. DEM interpolating	244
7.8.6. DEM cutting by polygons	245
7.8.7. Convert DEM to points	247
7.8.8. Adding of points to DEM	249
7.8.9. The DEM editing toolbar	250
7.8.10. Elevation correction along axis line	252
7.9. Accuracy control of DEM creation	257
7.9.1. Creation of difference DEM	257
7.9.2. DEMs comparing	259
7.9.3. Search for DEM areas	261
7.9.4. DEM control by TIN	264
7.9.5. DEM control by vector objects	266
7.9.6. DEM control by triangulation points	269
7.9.7. Creation of histogram by DEM	271
7.9.8. Creating statistics file by DEM	274
7.9.9. Lists of DEM min/max values	275
7.9.10. Subtracting the matrix values from the constant height	277
7.9.11. Slopes map creation	278
7.9.12. Building DEM elevation profile along a linear object	281
7.9.13. Assigning extreme DEM heights within the polygon as attributes	283
7.10. DEM saving	284
7.11. DEM recovering	285
7.12. DEM loading	286
7.13. DEM export	287
7.13.1. DEM export	287
7.13.2. Export to CSV	288
7.13.3. Export to DTED	289
7.13.4. Export to ERDAS Imagine	290
7.13.5. Export to GeoTIFF	291
7.13.6. Export to PCIDSK	293
7.13.7. Export to MTW	295
7.13.8. Batch DEM export	296
7.13.9. Convert DEM to point cloud (LAS)	297
7.13.10. Convert DEM to point cloud (LAS) in batch mode	298
7.14. DEM import	301

7.14.1. Batch DEM import	302
7.14.2. Batch import from folder	303
8. Volumes Calculation	305

1. Purpose of the document

This document contains detailed information about generation of digital elevation model (DEM). The document describes creating of regular nodes grid and working with layers needed for DEM generation points layersf, triangulated irregular network (TIN), contour lines and DEMs. The document also describes main ways of generation of digital terrain model using these layers.

2. DTM Generation

The system provides possibility of generation of digital terrain model using different data sets.

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.

It is possible to acquire data necessary for DTM generation both from central projection images and from satellite scanner images.

The **Terrain** menu is used for working with the above mentioned data sets.

2.1. The “Terrain” menu

Table 1. 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

Menu items	Function
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	to calculate a volume (i.e. <i>embankment</i> or <i>excavation</i>) 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.
Contours	The Contours menu contains usual menu items used for contours generating, editing, import and export
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)
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)

3. Regular grid of nodes

3.1. Grid creation


The system provides possibility of regular *grid* creation with specified step. The grid is used for points creation in automatic mode with help of correlator or in semi-automatic pathway mode. A grid step is used as a frequency of nodes creation, in which vicinity spatial coordinates are calculated and points are created. Moreover, the grid is used for different operations, such as map creation in stereo quality, DEM generation using dense model and for other operations (see the chapters [Automatic points creation](#), [Points creation in pathway mode](#) in the “[Vectorization](#)” User Manual).

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.



It is possible to use [pre-regions](#) for grid area detection.

- 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.

To save created grid in active profile resources use **Grid › Save** and **Grid › Save as** menu items. The grid is saved to the *.x-grid file.

To open existing grid, select **Grid › Open**. To close a grid, select **Grid › Close**. To close all grid layers, select **Grid › Close all opened layers**.

Right-click menu of the *Grid* layer in the *Manager* duplicates save and load commands in the **Grid** menu, and allows to close a grid layer.

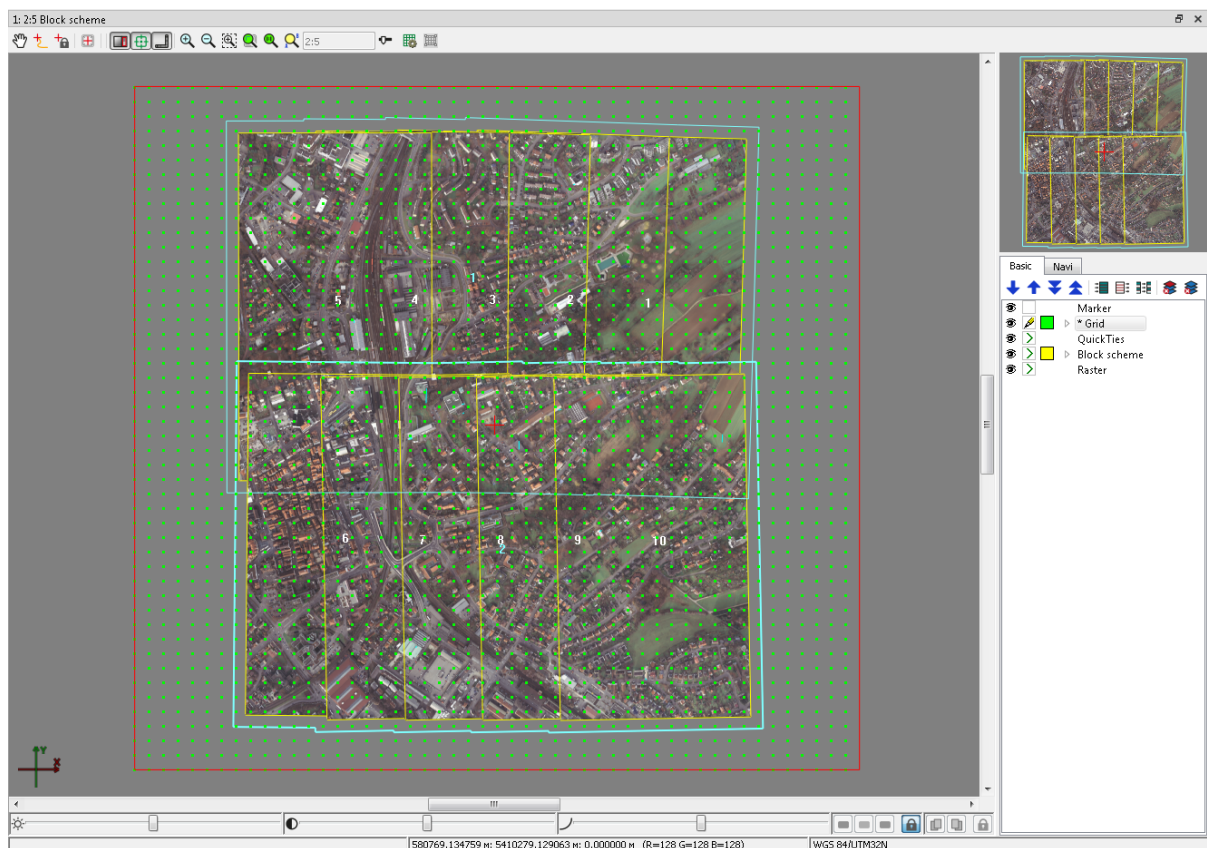


Fig. 1. Rectangular grid for the whole images block

3.2. Grid parameters

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

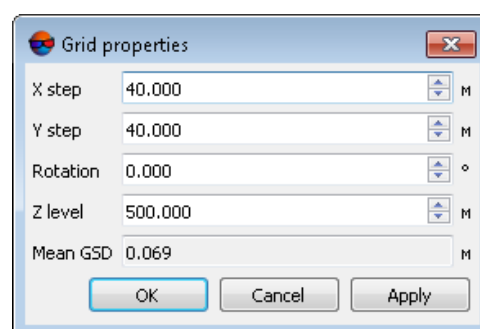


Fig. 2. Regular grid parameters

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;



To change a direction of grid nodes tracking in the **pathway mode** specify negative values of grid step.

- **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.

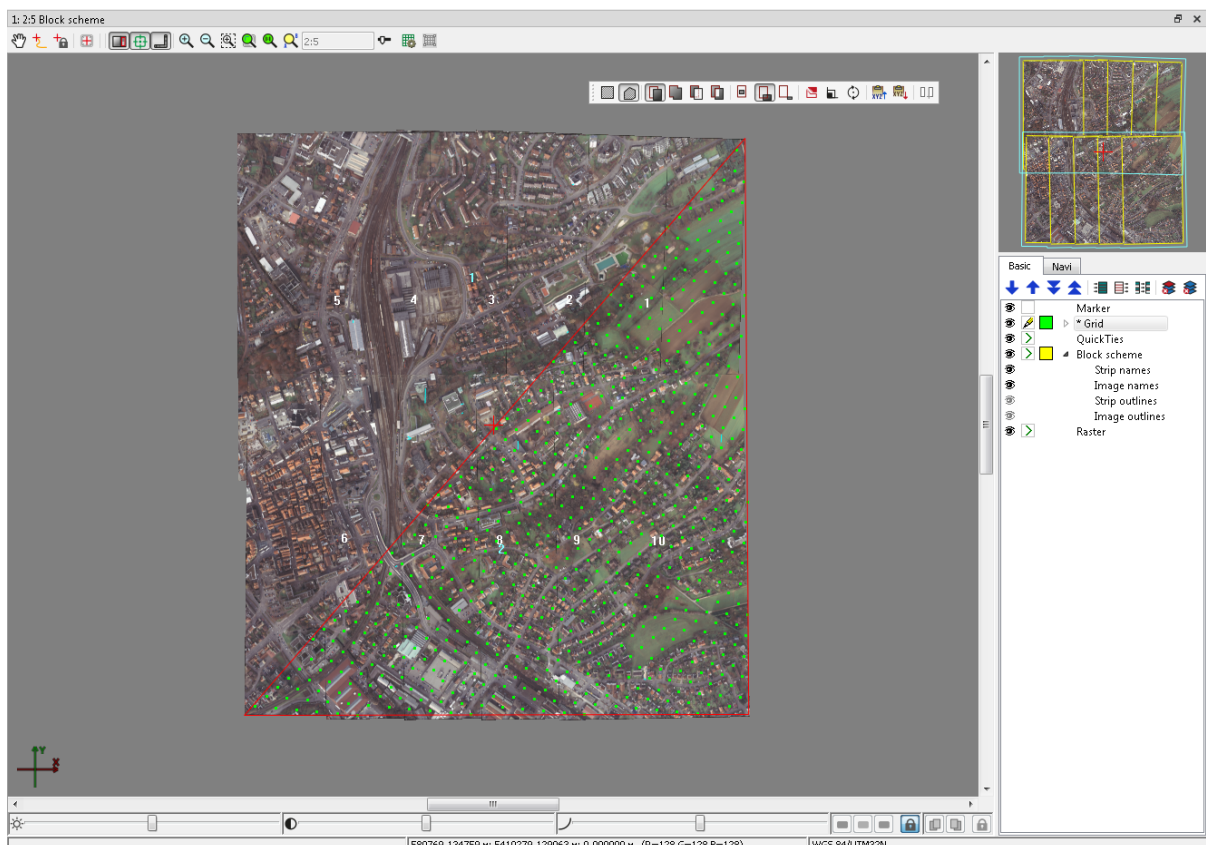


Fig. 3. Grid created using arbitrary border (with grid rotation)

In order to change grid display parameters double-click on layer objects color button (□) in a *Layer manager*. The **Layer display settings** window opens. It allows to setup color of grid nodes and borders, as well as a type and size of symbol used to show grid nodes.

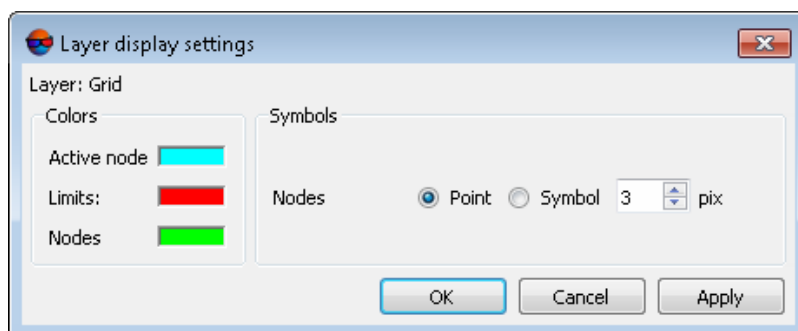


Fig. 4. Parameters of grid display

Additional parameters of grid display, as well as settings of pathway mode are setup on the **Grid** tab of the **Settings** window (see the '[General system's parameters](#)' User Manual).

4. Points

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



4.1. The “Points” menu


The **Points** menu contains standard menu items used to load, save, import and export *vector layers*, as well as to perform specific operations with *point objects* (creation and filtering).



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

Table 2. Brief description of the 'Points' menu

The Points menu	Function
 Load (Ctrl+O, V)	to load <i>vector</i> layer from *.x-data file
Recent	to perform the quick access to recently loaded <i>vector</i> layers
Save	to save or rewrite active <i>vector</i> layer as file with *.x-data extension
Save as...	to save active <i>vector</i> layer with new name as file with *.x-data extension
Save selected as...	to save just selected <i>vector</i> objects
 Close	to close <i>vector</i> layer

The Points menu	Function
 Compute points automatically	to <i>calculate points automatically</i> using correlator in overlap areas of stereopair images by regular nodes grid
Pathway	contains items used for <i>semi-automatic points creation</i> using regular grid
Filter	contains items <i>used</i> for deletion, correction or recognizing <i>points</i> which have fallen on buildings, trees, cars, in pits, etc. during automatic points calculation, and for filtering of accident surges
Import	contains items used to import <i>vector</i> layers from files with different extensions (see the “Import of vector objects” chapter of the “ <i>Vectorization</i> ” User Manual)
Export	contains menu items used to perform export of <i>vector</i> layers to different formats (see the “Export of vector objects” chapter of the “ <i>Vectorization</i> ” User Manual)

4.2. Points creation

4.2.1. Points creation modes and methods

The system allows the following modes of points creation:

- *manually in stereomode* – points are added manually in stereomode using correlator or without it;



Correlator's parameters are setup in the window of general system's settings on the **Correlator** tab (see the “*General system's parameters*” User Manual).

- in *semi-automatic pathway mode* with preliminary creation of regular nodes *grid* – that means sequenced passing of all nodes of regular grid and adding of points manually using correlator or without it, or skipping a node and passing to the next one;



Correlator's parameters are setup in the window of general system's settings on the **Correlator** tab (see the “*General system's parameters*” User Manual).

- in *automatic mode* with preliminary creation of regular nodes *grid*. At that in grid nodes vicinity the system calculates points spatial coordinates with help of correlator and creates points if the coordinates are calculated successfully. In automatic mode the system passes grid's nodes specified number of times with different correlator parameters, performs accuracy control, points filtering, and saves quality evaluation.











For automatic points calculation correlator's parameters should be adjusted in the **Compute points** window.

4.2.2. The “Pathway mode” toolbar

For work in pathway mode the system provides the **Terrain › Points › Pathway mode** menu items, that are partially duplicated by the **Pathway mode** additional toolbar buttons.

Table 3. The 'Pathway mode' toolbar

Buttons	Function
 Activate	to turn on/off pathway mode
Add point (Enter)	to add point to current node
Bypass point (Delete)	to bypass a node (without adding point) and to pass to the next node
Return (Backspace)	to return to the previous node (without point deletion)
 Marker to current grid node	to move marker to current active grid node
 To nearest grid node	to move marker to nearest grid node of markers current position. The node becomes active automatically (provided that marker is inside grid in geodetic coordinate system), that allows to start passing points manually
 To first grid node	to move marker to left lower grid node and to make the node active
 Mode: scan rows	to turn on sequential mode of grid passing by rows
 Mode: scan columns	to turn on sequential mode of grid passing by columns
 Mode: 'zigzag' by rows	to turn on 'zigzag' mode of grid passing by rows
 Mode: 'zigzag' by columns	to turn on 'zigzag' mode of grid passing by columns
Settings	opens the system settings window on the Pathway mode tab to setup the mode parameters (see Section 4.2.4)

4.2.3. Points creation in pathway mode

For *semi-automatic points creation* using regular grid the system provides **Pathway mode**.

In the pathway mode the system passes grid nodes in such a way that XY coordinates of points are measured uniformly. Marker XY coordinates are preserved while passing from one grid node to another, and elevation is measured manually. Points measurement in pathway mode is possible both using correlator or without it.

The first coordinate of marker elevation during passing to the first node corresponds to grid level by Z, specified in the [Grid properties](#). Marker's elevation level during passing to each next node is defined by settings.

Points adding in pathway mode is performed in stereomode in stereopair 2D-window.



If a grid is created for the entire images block, the system allows to setup automatic pass to the next stereopair while passing nodes. To do this use the window of general [system's parameters](#) on the **Pathway** tab.

To create points in pathway mode, perform the following actions:

1. [Create a grid](#) for required area of points creating.
2. Create or make active a vector layer for points creation.
3. Choose **Terrain › Points › Pathway › Activate** or click the button of the **Pathway mode** additional toolbar. Marker will be placed to left lower grid corner – start point of passing.



If grid node was not created before, it is created automatically for the whole images block.



When the grid layer is closed the **pathway mode** is turned off automatically.

4. Select a direction of grid nodes passing using **Terrain › Points › Pathway** menu or using buttons of the **Pathway mode** additional toolbar.
5. Choose **Window › New 2D-window (stereopair) (Ctrl+Alt+L)** or click the button of the main toolbar. The stereopair 2D-window is opened and allows to start grid nodes passing.
6. [optional] For image automatic centering by marker during passing to the next grid node turn on the **Fixed marker mode**. To do this click the button or use the **F6** hotkey.



To vectorize objects during grid nodes passing in pathway mode, turn the **Fixed marker mode** temporarily off.

7. Click the to turn on stereo mode in 2D-window of a stereopair.



To work in pathway mode it is necessary to turn on the mode of point objects creation – using the button of the **Vectors** additional toolbar.

8. To perform grid nodes passing the system provides the following tools:
 - to add point to current grid node manually, set marker elevation using mouse wheel and press **Enter**;
 - to add point to current grid node using correlator, press **Space** key to set marker elevation using correlator and press **Enter**;



After the point is added marker moves to the next grid node according to selected passing direction.

- to pass to the next node without adding point, press **Delete** key;
- to pass to the previous node without adding point, press **Backspace** key.





To delete point use the **Ctrl+Z** hotkeys. It is possible to delete points, only if history of recent operations is turned on and cancelling of these operations is allowed (see the *Undo settings* chapter in the “[General system's parameters](#)” User Manual).

9. After passing of the last grid node the system displays the Pathway complete message. Click OK. Save the vector layer with acquired points set for further use.



To memorize a position of the current active node during passing, save the grid layer.

To continue grid nodes passing from current position, perform the following actions:

1. Load saved grid layer.
2. Choose **Terrain › Points › Pathway › Activate** or click the  button of the **Pathway mode** additional toolbar. After that a grid node which was active at the moment of the layer saving, becomes active automatically.
3. Place marker to vicinity of the active grid node.
4. Choose **Terrain › Points › Pathway › To nearest grid node** or click the  button of the **Pathway mode** additional toolbar, to move marker to the nearest node.
5. Go on to pass grid nodes.

4.2.4. Parameters of pathway mode

To specify preliminary parameters of work in pathway mode, perform the following actions:

1. Choose **Terrain › Points › Pathway › Settings**. The **Settings** window is opened on the **Pathway** tab (see the detailed description of general system settings in the “[General system's parameters](#)” User Manual).

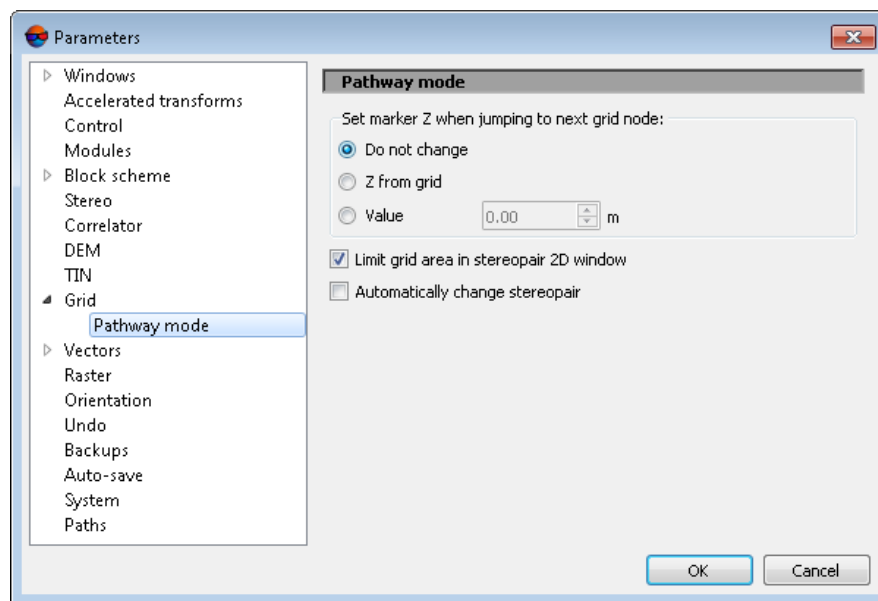


Fig. 5. Parameters of pathway mode

2. Choose one of the ways of specifying marker position by Z when pass to next grid node:
 - **Do not change** – marker position by Z is not change, i.e. Z-coordinate of marker in previous grid node is used;
 - **Z from grid** – marker position by Z is set equal to Z level of grid, defined in the **Grid properties** window;
 - **Value** – marker position by Z is set equal to the value specified in this field (in meters).
3. [optional] The **Limit grid area in stereopair 2D-window** checkbox is set on in the system by default, that allows to setup passing grid nodes in pathway mode only in borders of stereopair regardless of grid size.



It is not recommended to set the **Automatically change stereopair** checkbox on while the **Limit grid area in stereopair 2D-window** checkbox is set on, since it leads to work with incorrect data.

4. [optional] In order to pass nodes of 'global' grid with automatic stereopair change, set the **Limit grid area in stereopair 2D-window** checkbox off and set the **Automatically change stereopair** checkbox on. Besides, the **Threshold** parameter also has influence on automatic stereopair change. The parameter specifies a distance from stereopair border in percent of the whole stereopair size, and when this distance is reached during passing nodes in pathway mode the stereopair is changed to the next one in 2D-window.

4.3. Points automatic calculating

4.3.1. Automatic calculating of points

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



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.

For the satellite scanner imagery projects, the **stereoscopic processing** is to be carried out during the adjustment. (see the “Adjustment of scanner blocks” chapter of the “[Block adjustment](#)” User Manual).

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.

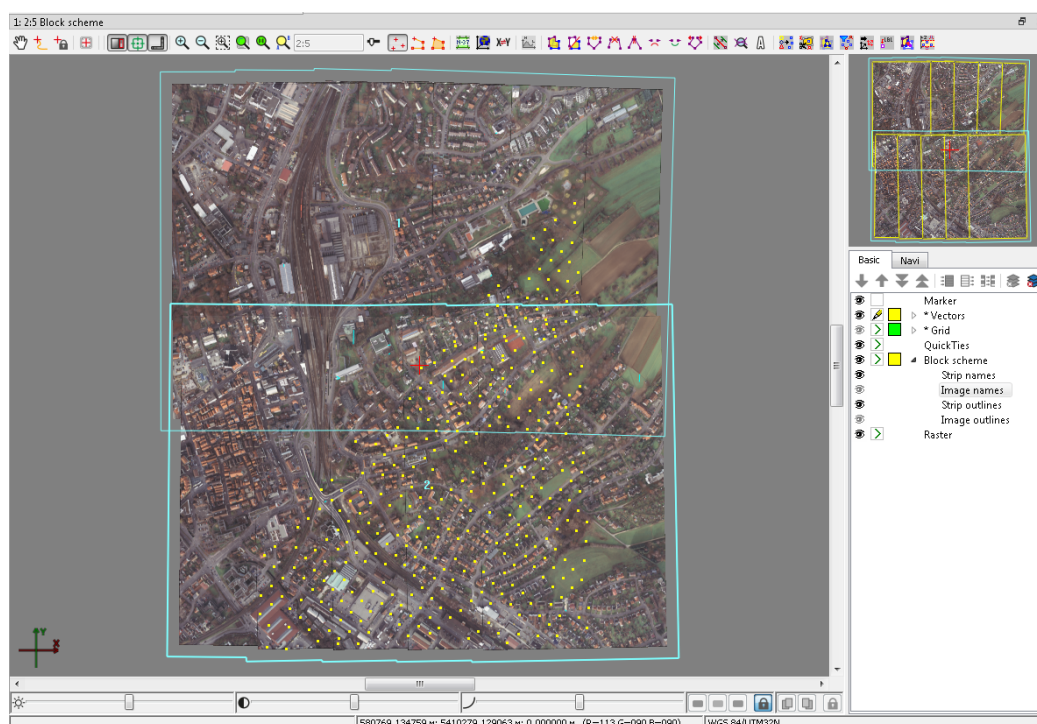


Fig. 6. Automatic points calculation in arbitrary area

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**. 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.

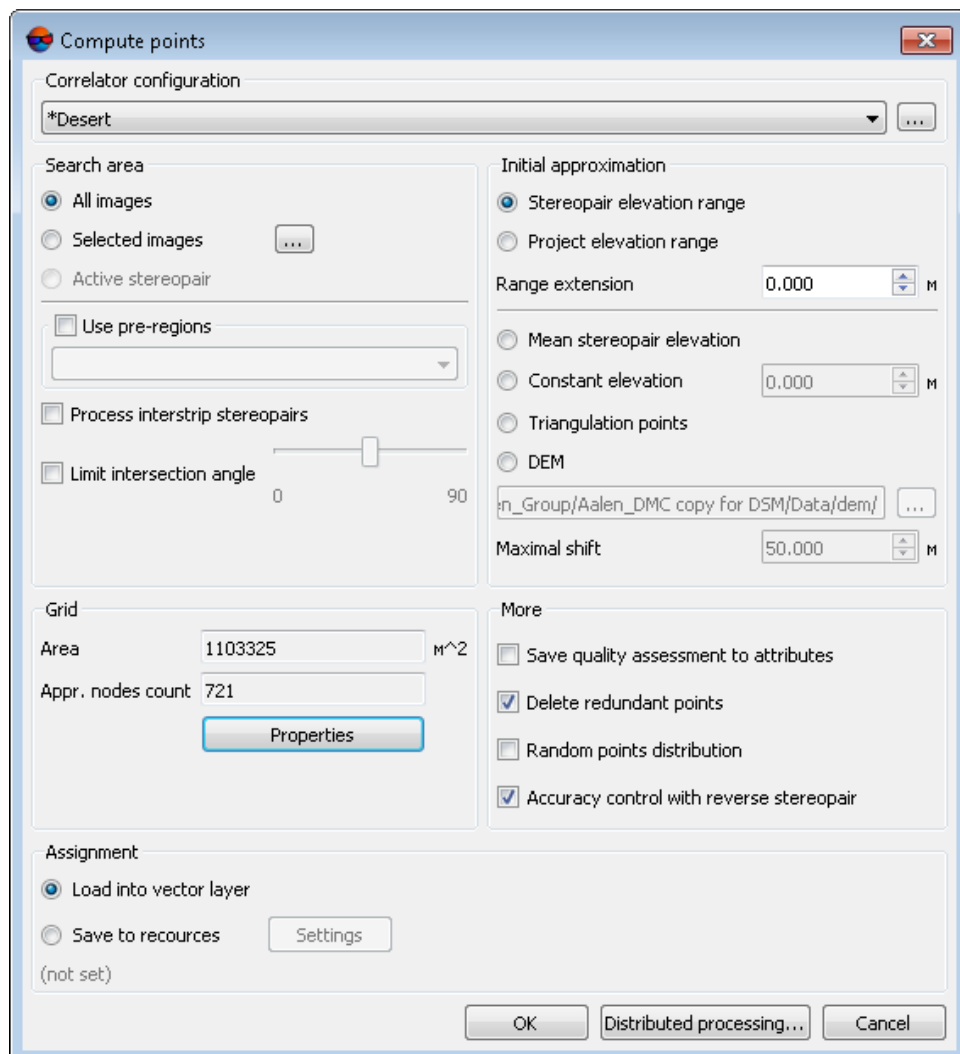



Fig. 7. The Compute points window

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.

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.



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.

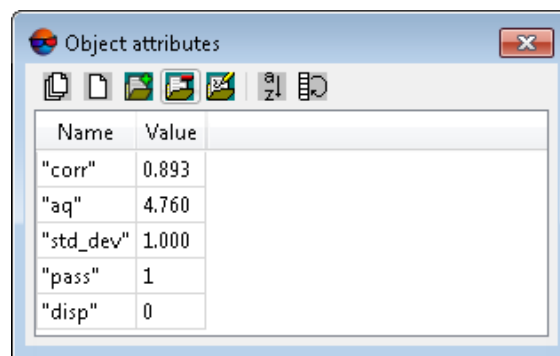


Fig. 8. Quality assessments of calculated point

- **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.

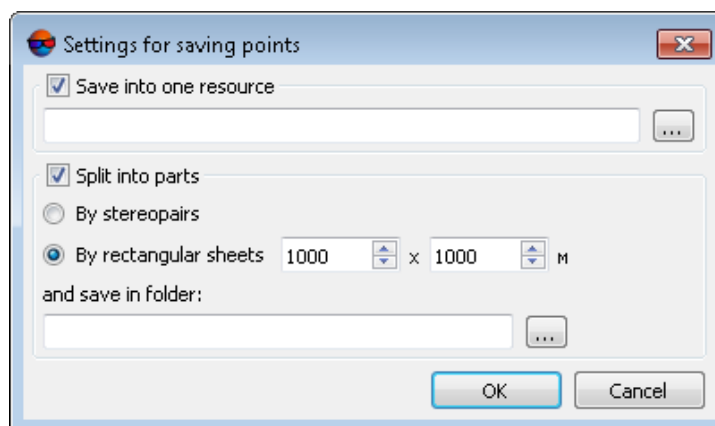




Fig. 9. The Settings for saving points window

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.

4.3.2. Correlator configurations

Correlator configuration – is a script of points calculation operation, that defined a number of passes of regular grid nodes and a set of correlator parameters for each pass.

The **Correlator configuration** window is used to form a list of correlator's configurations. In order to open the window click the  button in the **Correlator configuration** panel of the **Compute points** window.

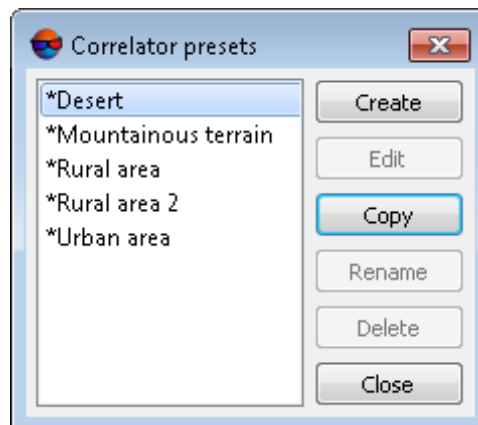


Fig. 10. The 'Correlator presets' window

The left part of the window displays a list of all created correlator configurations. Default list includes standard correlator configurations with fitted values of parameters for the following terrain types:

- *mountainous terrain* – for automatic points calculation on images with mainly mountainous terrain with big elevation differences;

- *urban areas* – for automatic points calculation on images with mainly urban type of build-up;
- *desert* – for automatic points calculation on images with monotonously deserted terrain with rare vegetation;
- *rural area* – for automatic points calculation on images with rural areas with with minor amounts of buildings and infrastructure;
- *rural area 2* – for automatic points calculation for 'rural' area with heterogeneous parts of relief. If there are images with big intersection angle the configuration uses a mask of larger size. Accuracy of points calculation is less then when using *rural area type*.

The right panel of the window contains the following buttons for managing correlator's presets:

- **Create new** – allows to specify name and parameters to [create new configuration](#) of correlator;
- **Edit** – allows to edit parameters of correlator's configuration selected in the list;
- **Copy** – allows to create a copy of selected correlator's configuration with different name, but with the same parameters;
- **Rename** – allows to rename selected correlator's configuration;
- **Delete** – allows to delete selected correlator's configuration;



Preset configuration couldn't be changed or removed.

- **Close** – allows to close the **Correlator presets** window.

4.3.3. Setup of correlator configuration

To view and setup correlator's configuration the system provides the **Edit preset** window, which is opened in the following cases:

- while editing of existing configuration select configuration in the list of the **Correlator presets** window and click the **Edit** button;
- while creating new configuration click the **Create new** button, input new configuration name and click OK.

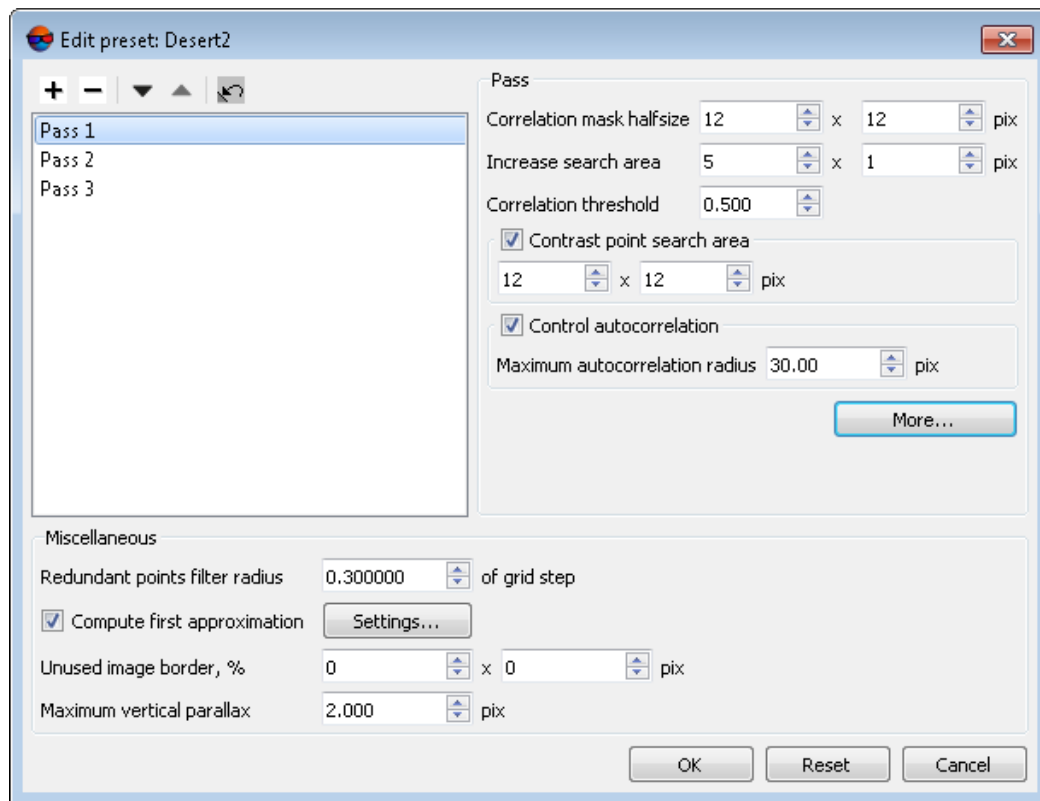


Fig. 11. Window of correlator passes parameters

Setup of correlator configuration in the **Edit preset** window includes defining of script of automatic points calculation.



Depending on terrain type the script could include both single and multiple passes of grid nodes, it is possible to specify particular set of correlator parameters for each of them.

For correlator configuration setup perform the following actions:

1. Create a list with required number of correlator passes.



Passes in the list are placed according to their order of use during automatic points calculation.

To form the passes list the following buttons are provided:

- **+** – allows to add new pass to the end of list;
- **-** – allows to delete selected pass;
- **▼** – allows to move selected pass down the list;
- **▲** – allows to move selected pass up the list;
- **↺** – allows to return settings to default.

2. Select correlator pass in the list to setup its parameters.
3. In the **Pass** section setup the following main correlator parameters for a pass selected in the list:

- **Correlation mask halfsize** – half of linear size of rectangular correlation mask in pixels by X and Y axes;



Increasing the mask size increases recognition quality for smooth relief and decreases influence of tiny details on terrain surface, however, it decreases total amount of acquired points. Besides, during processing imagery with low-contrast areas, which correspond to sloping grounds (fields or lawns on hill slopes, for instance), the system may add points that correspond an edge of mask, rather than center.

- The **Increase search area** parameter allows to specify a number of pixels to increase search area more than defined value of correlation mask size;



Increasing of search area is used, when considering stereopair orientation data the estimated search area of corresponding point on the right images is insufficient (no tie points enough on high-rise objects or in stereopair corners, for instance). It is necessary to increase search area by Y in case of poor accuracy of stereopair relative orientation.

- **Correlation threshold** – minimal acceptable value of correlation coefficient;



Value of correlation threshold depends significantly both on material and other parameters. For correct work of correlation algorithm it is recommended to setup the value not less than 0.7.

- **Contrast point search area** – a size of area around grid node, where the system performs search for a point with maximal contrast prior to correlation operation;



It is recommended to increase this value for images with well-defined contours of objects, and to decrease it for low-contrast (fuzzy) areas, where there is high probability of correlator errors.

- **Control autocorrelation** and **Maximum autocorrelation radius** – is used to control point's autocorrelation – degree of point's uniqueness in some its vicinity on the left image.



The more the autocorrelation radius, the less distinctive is the point and the more probable is incorrect matching of the point with the right image, including cases with high correlation coefficient.

The **More** button is used to setup [additional parameters](#) of selected pass.

4. In the **Miscellaneous** section setup the following additional parameters of correlator's pass:

- **Redundant points filter radius** – is used to filter nearest points, acquired during passing the same grid nodes on different stereopairs;



Points thinning is performed after completing of points calculation operation, considering specified value of the radius in meters.

- **Compute first approximation** – allows to use results of preliminary points calculation, which parameters are setup using the **Settings** button;



It is recommended to perform preliminary calculation for automatic search for points on satellite scanner images.

- **Unused image border** (in percent) – allows to not consider specified percent of image edges during points calculation;
- **Maximum vertical parallax** – allows to specify a limit for points filtering by vertical parallax value (in pixels).



Point acquired as a result of correlation has 4 coordinates – (XL, YL) on the left image and (XR, YR) on the right image. To calculate vertical parallax the system performs transformation of images coordinate system into project coordinate system and vice versa. As a result two pairs of new coordinates are acquired (XL_2, YL_2) and (XR_2, YR_2) . In this case parallax is calculated as $Y_{PAR} = (YR - YL) - (YR_2 - YL_2)$. Points with the value more than specified Y_{PAR} are rejected. This parameter should be selected experimentally, especially for scanner survey projects.

5. Click OK to complete setup of correlator configuration parameters.

4.3.4. Additional correlator's parameters

To setup additional parameters of correlator for selected pass in the **Edit preset** window there is the **More** button, that allows to open the **Additional settings:** window **Pass #**.

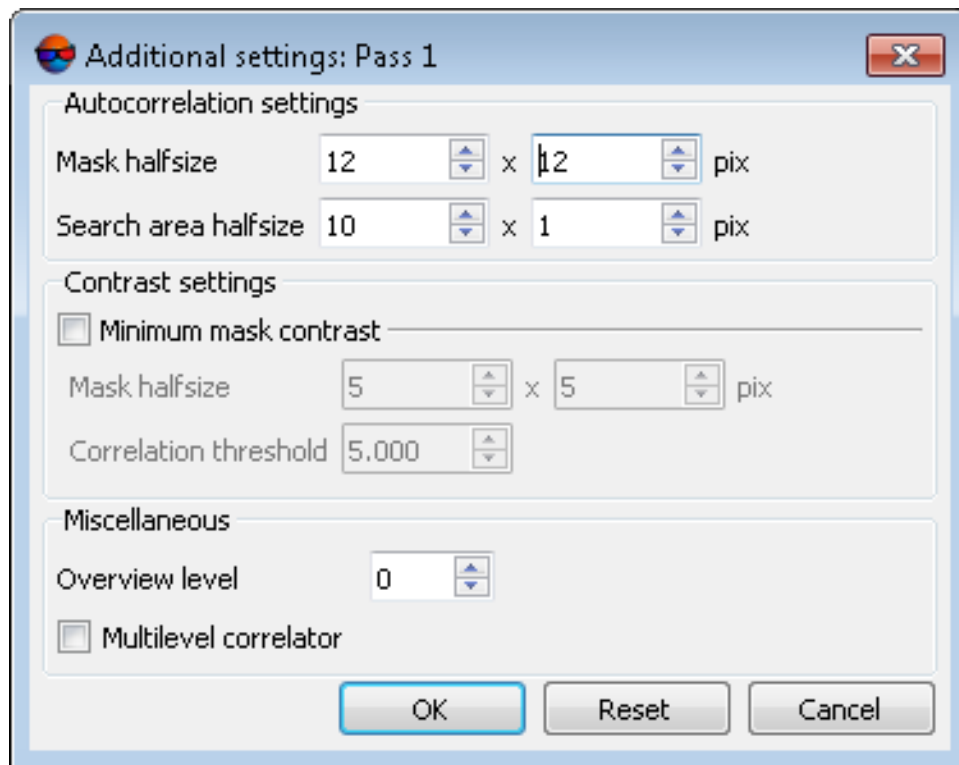


Fig. 12. Additional parameters of selected pass

The **Autocorrelation settings** section allows to specify additional parameters of auto-correlation:

- **Mask halfsize** – half of linear size of mask for auto-correlation in pixels by X and Y axes. Default value – 10x10 pixels;
- **Search area halfsize** – a halfsize of area, where auto-correlation surface is created.

The **Contrast settings** section allows to consider contrast for low contrast images. Set the **Minimum mask contrast** checkbox on and specify the following parameters:

- **Mask halfsize** – a halfsize of area around a point, where the system performs contrast calculation;
- **Minimum contrast** – value lies in range from 0 to 100.

The **Overview level** parameter allows to perform correlation between thinned images and is used in the following cases:



'0' value corresponds to initial image, in other cases that system uses the level of the pyramid with a specified number.

- for film data with coarse grain, which prevent from reliable operation of correlator;

- on relief surface like fields, that in initial detalization have occasional irregular items (grass, small bushes), or regular pattern (new-ploughed field). When zoom out such a terrain surface, the textures comparable on the left and right images appear;
- to exclude small objects on relief (bushes, separate trees, buildings). In this case, the thinning together with increasing of mask size leads to the fact that even if the center of the mask falls onto the protruding object, the resulting Z-coordinate of point corresponds to the surrounding terrain elevation.

As the passes of the same calculation process, it is advisable to use a correlator configuration differing by no more than one level of the pyramid (for example, 0 и 1; 2 и 3). Otherwise, in areas of fine detail (in particular, buildings), it is possible to receive the 'chess' grids of alternating points at the height of the object, and at the height of the relief.

The **Multilevel correlator** checkbox allows to use different levels of pyramid for correlation during search for corresponding points.

4.3.5. Parameters of first approximation calculation

The system provides possibility to use first approximation of digital terrain model during correlator passes. Such DTM is a TIN, generated using thinned points set.



DTM of first approximation allows to significantly reduce the search area for each point of the main operation, i.e. to reduce the total search time and reduce the probability of erroneous points occurrence.



In correlator's preset configurations first approximation calculation is used for '*mountainous terrain*' configuration.

To use and configure calculation parameters of first approximation DTM, perform the following actions:

1. In the **Edit preset** window set the **Compute first approximation** checkbox on.
2. Click the **Settings** button. The **First approximation for:** window opens '**configuration name**'.

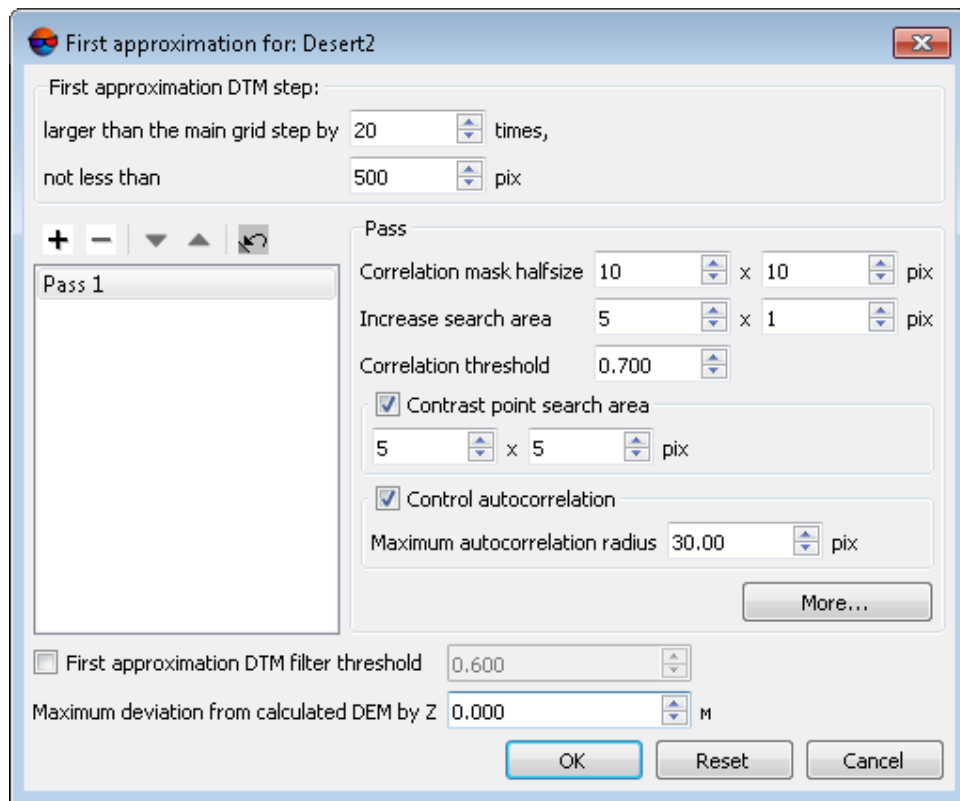


Fig. 13. Parameters of first approximation calculation

3. Specify the **First approximation DTM step** to define a step of DTM to be created.
4. [optional] To use threshold filtering during DTM creation set the **First approximation DTM filter threshold** checkbox on.

Specify a value that corresponds to threshold of [filtering](#) of intermediate TIN by minimum angle of inclination of the normal, given to a range of (0;1).

5. Set the **maximum deviation from calculated DEM by Z** in both directions.
6. The **Correlator passes** list and parameters in the **Pass** section allow to define correlator configuration for preliminary points calculation (see a description of the main parameters in the [Section 4.3.3](#)).

The **More** button allows to setup [additional parameters of selected pass](#).

7. Click OK to return to the **Edit preset** window.

4.3.6. 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**. 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.

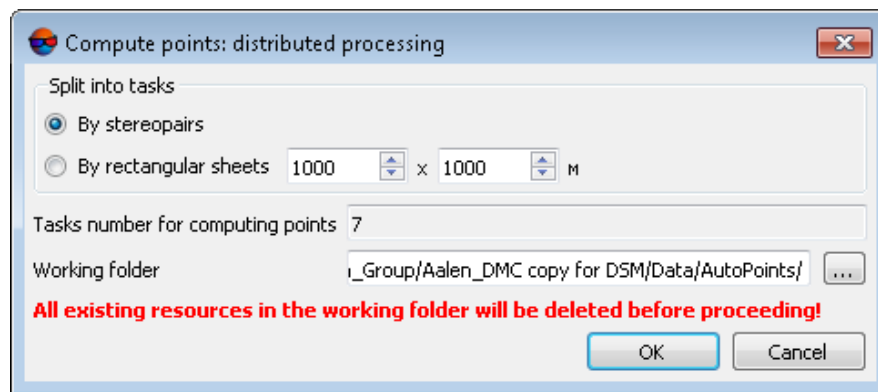


Fig. 14. Parameters of distributed processing of points calculation

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.


4.4. Points editing

The system provides possibility of editing of single or multiple selected points. Since points calculated automatically are points, all operations of points editing are applicable to them.

To select vector objects manually in active vector layer the system provides the following features:

- mouse double click or the **S** key to highlight single polyline/polygon;
- mouse click to highlight point/vertex (place marker in point's vicinity in advance);
- selection mode  **Rectangle** to select vector objects inside a rectangle;
- selection mode  **Polygon** to select vector objects inside arbitrary polygon.

To highlight objects on active vector layer *inside a rectangle* choose **Edit › Group selection › Rectangle** or click the  button of the **Tools** toolbar. Press and hold the **Shift** key and drag a rectangle by mouse.

To highlight objects *inside arbitrary polygon* choose **Edit › Group selection › Polygon** or click the  button of the **Tools** toolbar. Press and hold the **Shift** key. Then make mouse click to create the first polygon vertex and a 'rubber line' directed to current cursor position. Input subsequent vertices by mouse clicks. To complete objects selection by polygon use mouse double click. To break off selection, press the **Esc** key.




For quick switching between **Polygon** and **Rectangle** selection modes there is context menu opened by the **Shift+F10** hotkeys.

To select all objects of active vector layer choose **Edit › Select all** or use the **Ctrl+A** hotkeys.



See detailed description of selection tools and modes in the 'Vector objects selection' chapter of the '[Vectorization](#)' User Manual.

To add a new point to active vector layer, perform the following actions:

1. Choose **Edit › Vectors create mode › Points (P)** or click the  button of the **Vectors** additional toolbar to turn on points creation mode.
2. Place marker to selected point on image in 2D-window.
3. Press the **Insert** key to add a point without correlator on specified elevation.

The system also allows to place marker on a model surface automatically using correlator. The **Space** key is used to do this.



If the correlator failed to work the **Status** panel displays the Bad point message and the system produces warning audio signal (see the '[General system's parameters](#)' User Manual).

Besides the system provides possibility to delete points, located nearer from linear vector objects than specified distance (see the '[Vectorization](#)' User Manual).

The system provides possibility of vector objects topology control, i.e. search for duplicated points or points coinciding with vector objects vertices. For this the **Vectors › Topology › Verify topology** menu item is used (see the chapter 'Topology verifying' of the '[Vectorization](#)' User Manual).

4.5. Points filtering

The system provides deletion, correction or recognizing points which have fallen on buildings, trees, cars, in pits, etc. during [automatic points calculation](#), and filtering of accident surges.

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

To edit automatically calculated points the system provides the following filters:

- **Points thinning** – thinning out of densely located points with specified level of thinning;
- **Filter by Z-range** – filtering points and vertices of polylines/polygons, which Z-coordinate is out of specified range;
- **Median Z filter** – filtering points and vertices of polylines/polygons using mask of specified size;
- **Filter adjacent point objects** – filtering of closely located points (located nearer than specified distance);
- **Buildings and vegetation filter** – filtering points, which have fallen on tall objects (buildings, trees) or in pits to obtain layer of points, describing only terrain relief;
- **Surface objects filter** – filtering points, which have fallen on particular tall objects or pits of specific size.
- **Filter by image properties** – filtering objects depending on parameters of raster image. During preparatory stage (called “learning”) an operator selects specific image areas manually, serving as a reference samples for the filtering operation.

The set of filters used for points editing partly coincides with vector object filtering instruments (**Vectors** › **Filter**, see the “Vector objects filtering” chapter of the “**Vectorization**” User Manual).

4.5.1. Points thinning

The system provides possibility of removing of some part of automatically calculated points, located densely or close to each other. For this there is points thinning feature.

To perform points thinning perform the following actions:

1. Select **Terrain** › **Points** › **Filter** › **Thinning**. The **Points thinning** window opens.

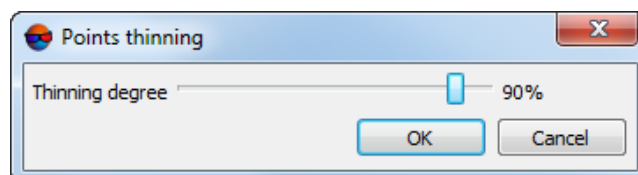


Fig. 15. Points thinning parameters

2. Specify degree of points thinning out in percent using the slider.

3. Click OK. After that the system removes specified percent of points, in order of their influence on quality of relief representation (the least significant points are removed first).

4.5.2. Points filtering by Z-range

The system provides possibility to remove points and vertices of polylines/polygons, which Z-coordinate falls outside specified range.

To perform objects filtering of active vector layer by elevations range, execute the following actions:

1. Select **Terrain › Points › Filter › Filter by Z-range**. The **Z-range filter** window opens.

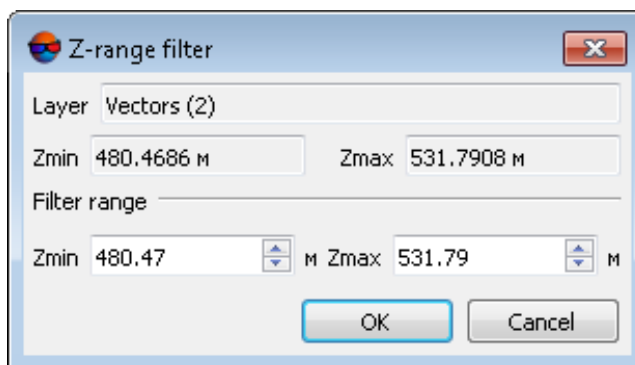


Fig. 16. Parameters of points filtering by Z

The **Layer** field displays a name of selected active vector layer, the **Zmin** and **Zmax** fields show terrain elevation difference in meters, calculated by all layer objects.

2. The **Filter range** section by default shows values of calculated terrain elevation difference. Specify maximal and minimal Z values in meters to consider for points filtering.
3. Click OK. Filtration removes all points and vertices of polylines/polygons, which Z-coordinate falls outside specified range. When the filtering operation is completed the system displays information message about number of deleted points.

Check and filtering by Z-range is also possible during [automatic points calculation](#).

4.5.3. Median points filtering by Z

The system provides possibility of median filtering to remove single sharp spikes in presence of smooth relief.

The *median filtering* contains the following sequence of actions:

1. Sequential scanning of area with vector objects by window-mask. Scanning step is specified as window-mask halfsize.
2. Deleting points which elevation is out of range.



The range is calculated as average level of vectors elevation that fall into mask. When calculating the range the system considers specified deviation from average level.

To execute median filtering perform the following actions:

1. Select **Terrain › Points › Filter › Median Z-filter**. The **Median Z filter** window opens.

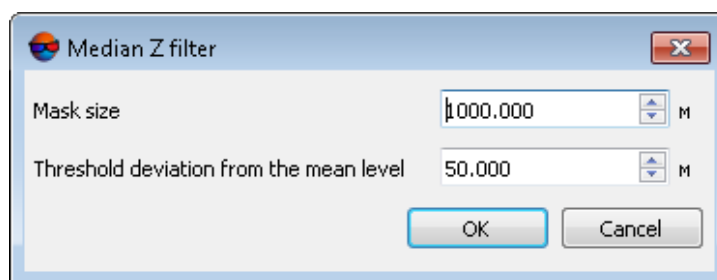


Fig. 17. Parameters of median points filtering

2. In the **Mask size** field specify a side size of scanning square window-mask in meters.
3. In the **Threshold deviation from the mean level** field specify deviation by Z in meters from mean level of objects elevation in the mask.
4. Click OK. Filtration removes all points and vertices of polylines/polygons, which Z-coordinate falls outside specified range. When the filtering operation is completed the system displays information message about number of deleted points.

4.5.4. Filter of adjacent point objects

Filter of adjacent point objects allows to remove points, in which vicinity there are other objects located closer specified distance.

To filter adjacent points on active layer perform the following actions:

1. Select **Terrain › Points › Filter › Filter adjacent point objects**. The **Filter adjacent point objects** window opens.

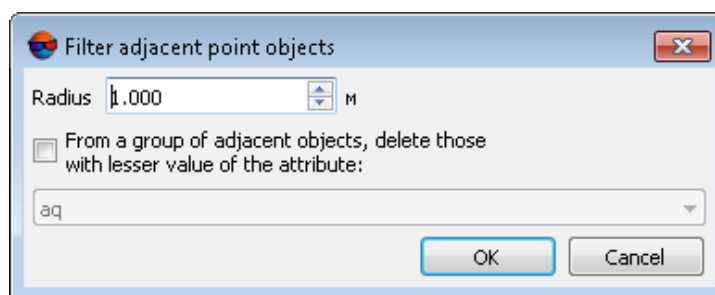


Fig. 18. Parameters of adjacent point objects filter

2. Specify the **Radius** – maximal distance in meters to each point, and if a points falls closer that the radius value, the points is deleted.
3. [optional] In order to delete from two points just one which contains the smallest value of one attribute (for example, if points vector layer was created in [automatic points calculation](#) mode with preserving of quality assessment in attributes), set the **From a group of adjacent objects, delete those with lesser value of the attribute** checkbox on and select in the list an attribute to be considered during points removal.
4. Click OK. Filtration removes points, between which the distance is less than specified. When the filtering operation is completed the system displays information message about number of deleted points.

4.5.5. 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.

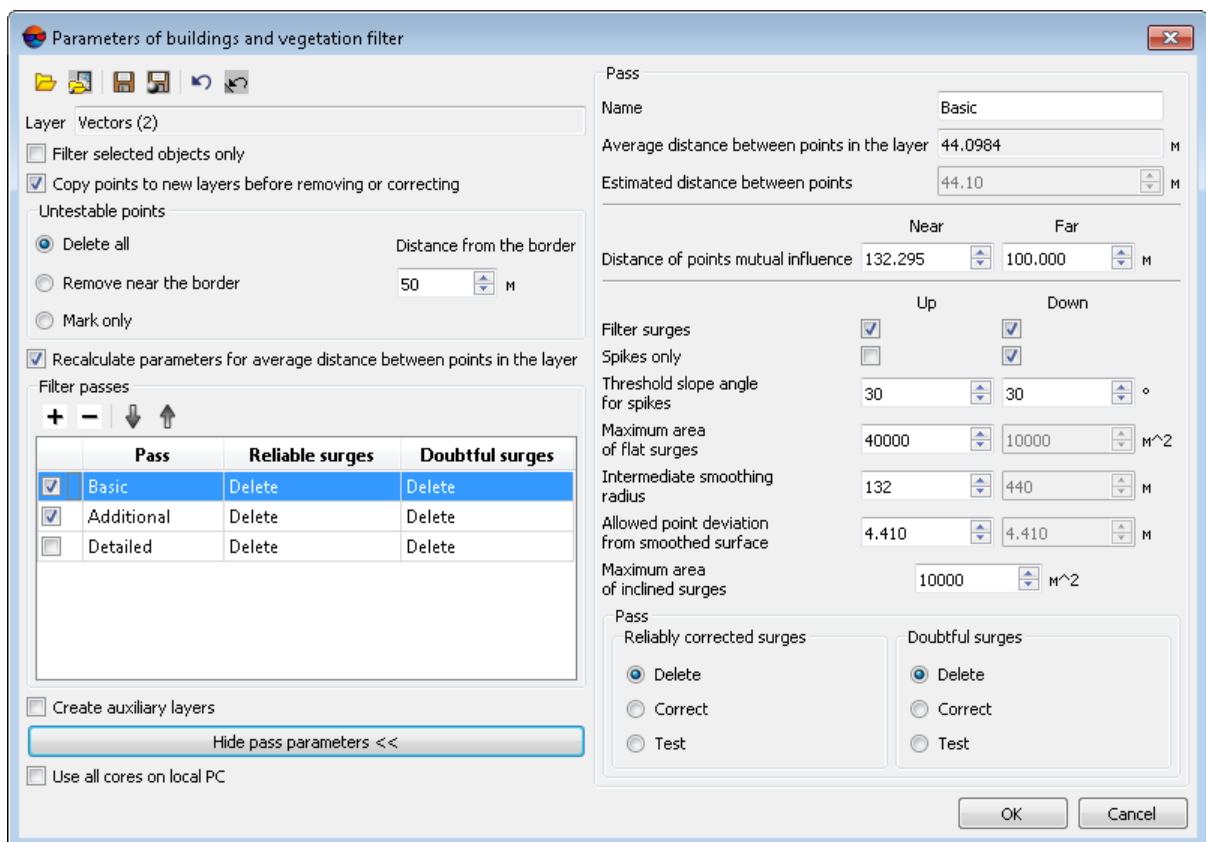








Fig. 19. Parameters of buildings and vegetation filter

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
	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**. The **Buildings and vegetation filter** window opens.

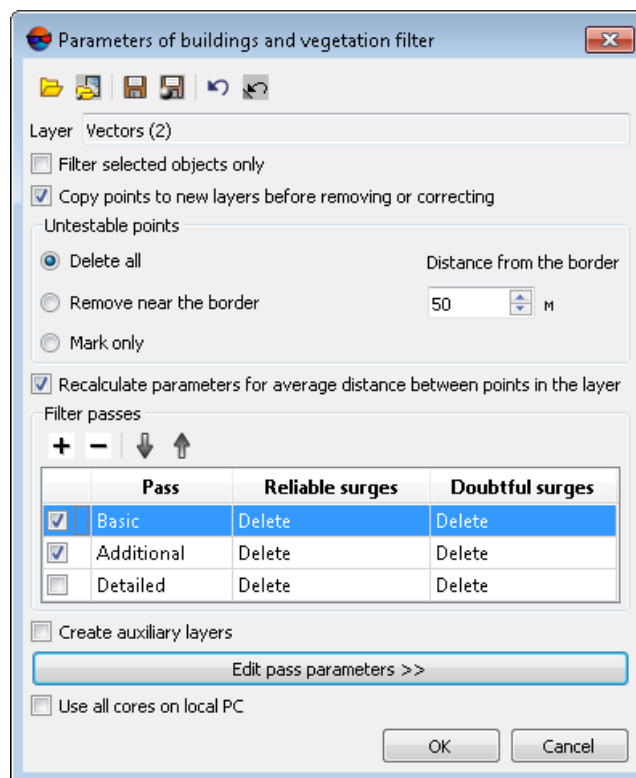


Fig. 20. Parameters of buildings and vegetation filter

- [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.

- [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:

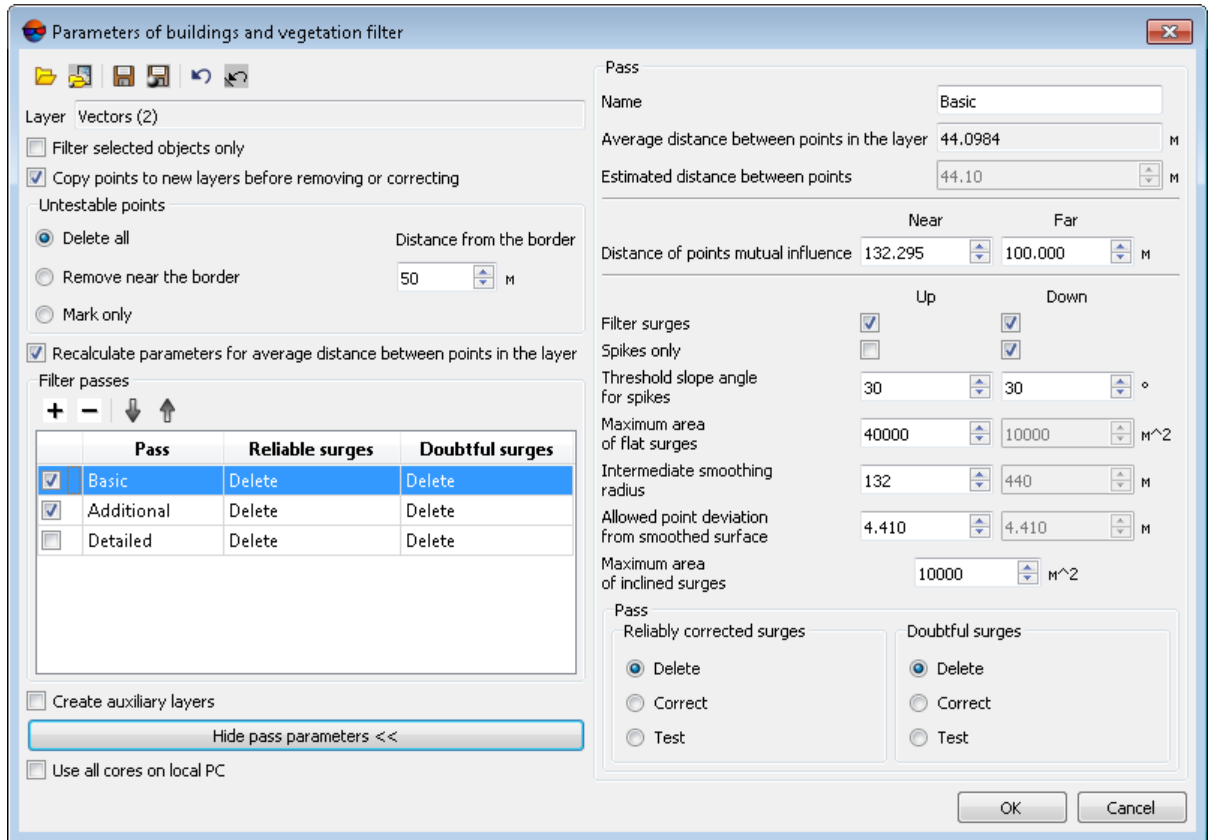


Fig. 21. Parameters of buildings and vegetation filter

- [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.

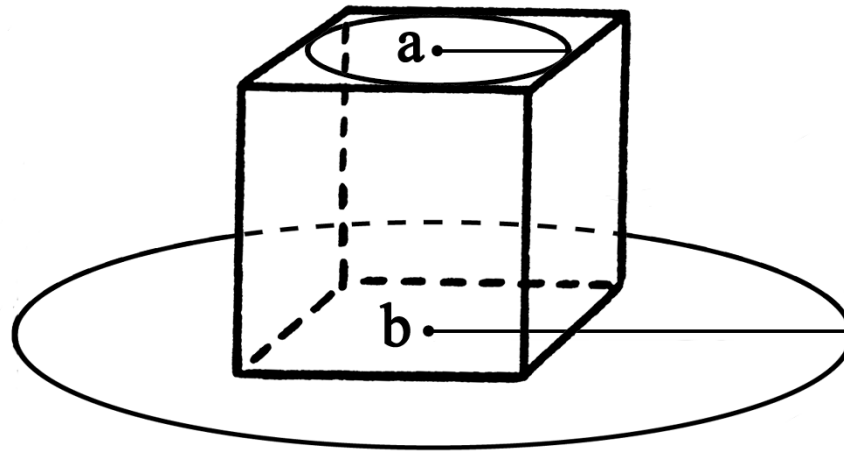


Fig. 22. Near distance of points mutual influence (a) and far distance of points mutual influence (b)

- 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 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.

4.5.6. Surface objects filter

The system provides possibility to correct blunders of correlation, obtained during [automatic points calculation](#). The filtering allows to delete points located on terrain surface with specified typical size by XY and Z.



This filtering type is not used for objects of completely built-up area, forest belts and similar objects.

The principle of filtering is to estimate the location and size of the contours created using TIN, which was generated with help of initial points. A surge is a group of contours that is not greater than specified size by XY and with elevations in specified range.



Surface objects filter is recommended to use to eliminate sharp spikes. For more fine filtering it is recommended to use this filter along with [buildings and vegetation filter](#).

To apply filter of objects on surface perform the following actions:

1. Select **Vectors › Filter › Surface objects filter**. The **Surface objects filter** window opens.

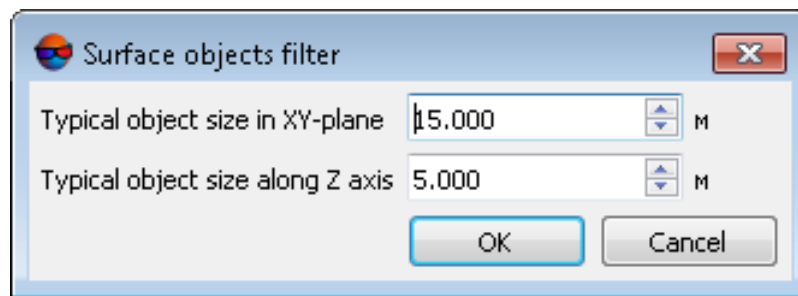



Fig. 23. Parameters of surface objects filter

2. Specify the **Typical object size in XY-plane** and **Typical object size along Z axis** in corresponding fields in meters.
3. Click OK to remove points found.

4.5.7. Filter by image properties

The system provides opportunity of objects filtering depending on parameters of raster image. During preparatory stage (called “learning”) an operator selects specific image areas manually (forested areas on example images), serving as a reference samples for the filtering operation.

To do filtering perform the following actions:

1. Make active a vector layer for filtering.
2. Select the raster image in 2D-window and choose **Window › New 2D-window (stereopair) (Ctrl+Alt+L)** or click the  button of the main toolbar. The stereopair 2D-window is opened and allows to start filtering.



The function is available in stereomode only.

3. Select **Terrain › Points › Filter › Filter by image properties**. The **Filter by image properties** window opens.

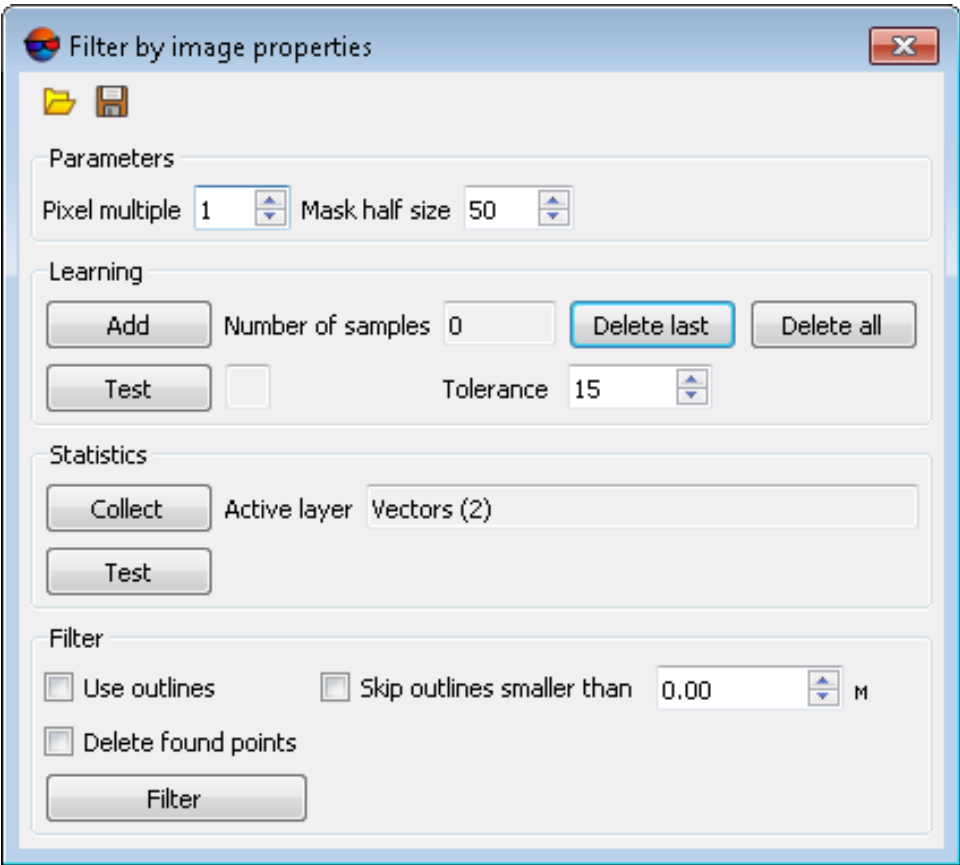




Fig. 24. Filter by image texture parameters

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 save a current script of filter work to *.x-filter resource located out of active profile resources

- 4. In **Parameters** section set the **Pixel multiple** and the **Mask half size** – a half size of the area around point where image characteristics are calculated;
- 5. Place marker over peculiar image area and click the **Add** button in the **Learning** section.



The **Delete last** button is used to remove the last selected sample, the **Delete all** button is used to remove all selected samples.

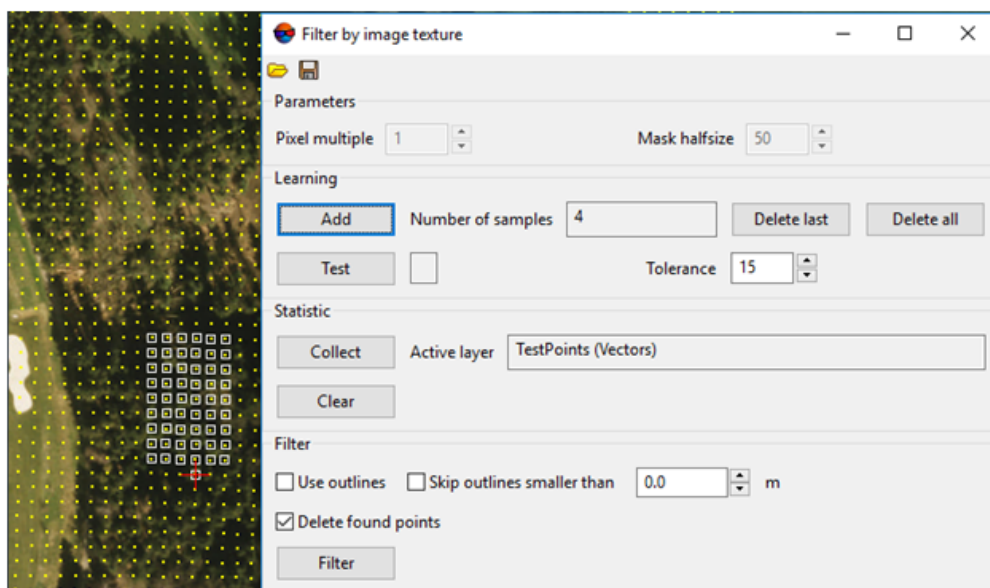


Fig. 25. Selection of the samples (filtering points in forestry area)

6. Repeat actions described in the section number **2** several times, using different marker positions.



To check training, place the marker on image area, which differs from the reference area (or vice versa - similar to it), and click the **Test** button. Status bar will show “-” or “+” result, that informs about match or mismatch of the test site characteristics with those obtained during training.

7. Click the **Collect** button in the **Statistic** section. Around every point inside of mask half size 6 statistic parameters of pixel brightness distribution are calculated for every colour channel. Statistic data are recorded in attributes of every point;



To look at statistics collected, select the point and select **Window > Objects attributes**.



To delete statistics collected from the objects attributes, click **Clear** button.

8. [optional] Turn on **Use outlines** checkbox to create polygons around areas containing filtered points automatically;
9. [optional] To create outlines only around large areas containing filtered points, turn on **Skip outlines smaller than** и задайте минимальный размера полигона в метрах;
10. [optional] Clear the **Delete found points** checkbox, if required.



While turning on **Delete found points** checkbox, filtered points will be automatically deleted after completion of filtration. While turning it off – automatically selected to perform further operations.

11. Click the **Filter** button.

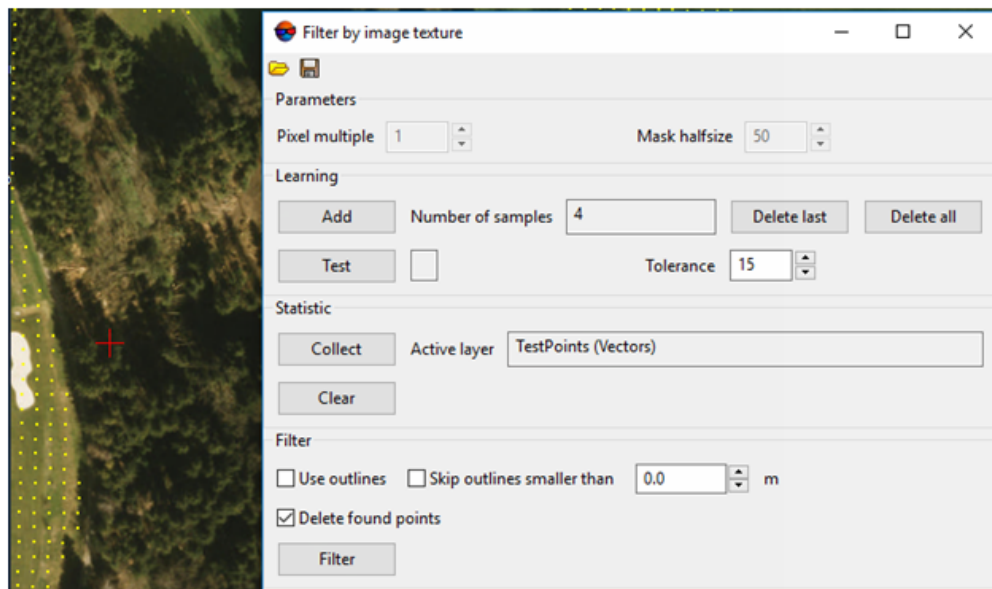


Fig. 26. Vector layer with points deleted after completion of filtering by image texture

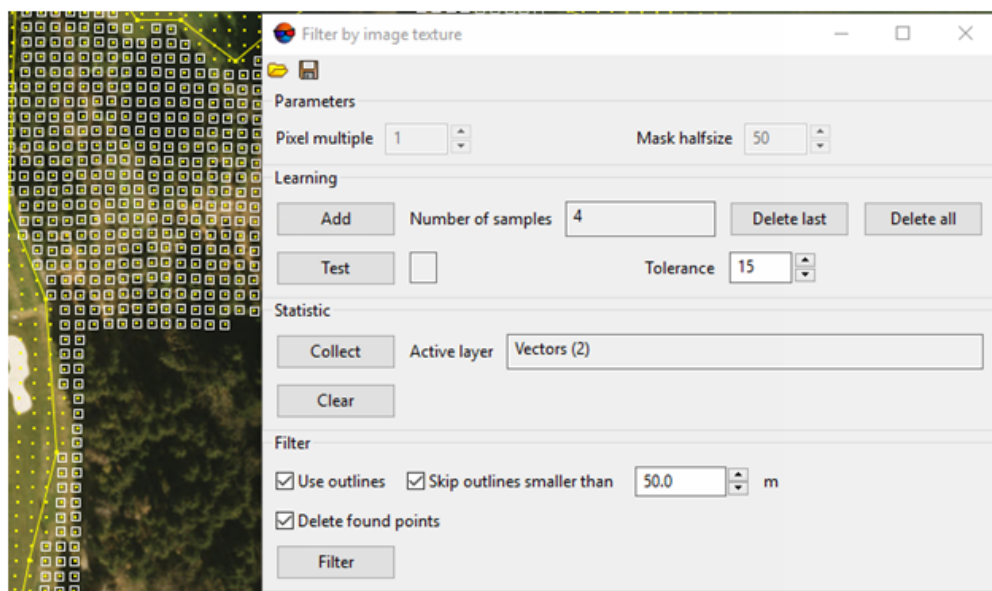


Fig. 27. Vector layer with points selected and outlines created after completion of filtering by image texture

5. Triangulation irregular network (TIN)



5.1. 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 4. Brief description of the 'TIN' menu

The TIN menu	Function
Load TIN... (Ctrl+O, T)	to load TIN from *.x-tin file
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
Save as...	to save active TIN layer with new name as file with *.x-tin extension
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 <i>Layer manager</i>
 Build (Ctrl+N, T)	to create TIN using loaded base layers
Build 3D-TIN...	to create textured 3D-TIN surfaces
Build 3D-TIN (Batch mode)	to create textured 3D-TIN surfaces in batch mode
Convert 3D-TIN CS	to change coordinate system of 3D-TIN
Restore...	to restore TIN from contours
Compute border	to create TIN border in automatic mode
 Rebuild	to rebuild TIN after editing of base layer objects
Check against adjustment points	to control TIN creation accuracy by triangulation points
Check against vector objects	to control TIN creation accuracy by vector objects , which were not used during TIN creation
Verify topology...	to check topology of TIN creation
The area of a polygon on the surface	to calculate TIN surface area within a polygon
Calculate area	to calculate TIN projection area on plane and area of TIN 3D surface
Statistic	to view statistic information about TIN
Simplify	to filtering of TIN triangles nodes using filtering of point vector objects in base layers, used for TIN creation

The TIN menu	Function
Peak filter	to peaks filtering – searching for TIN nodes, which Z value differs from adjacent ones significantly
Filter by Z-range	to edit TIN using Z-filter
Interpolate	to interpolate TIN to smooth DTM in order to build or enhance contour lines
Convert to vector layer	to convert TIN into a vector layer
 Embed objects	to embed vector objects layer to created TIN
Export	contains menu items to export results of TIN creation to DXF and CSV formats
Import	contains menu items allowing to import TIN from different formats
On/Off TIN visibility (Ctrl+T)	to enable/disable editable TIN layer visibility

5.2. General information

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.

The system also provides a possibility to build *quasi-contours* – isolines with specified step, which cross sides of TIN triangles. Quasi-contours are rebuilt automatically during editing of base layers containing vector objects. Quasi-contours are used to provide additional quality control of TIN generating and to reveal blunders.

During saving or loading TIN all base vector layers used for TIN creation are saved/loaded.



During TIN creation it is recommended to use breaklines – vector polylines along specific landforms (for example, watershed lines and valley-lines), as well as linear objects of human origin (for example, road-sides).

To perform TIN creation the system provides the following workflow:

1. [optional] [Pre-regions building](#).
2. Defining of set of [base vector layers](#).
3. [TIN borders creation](#).
4. [Building TIN](#).

The system provides the following features to work with TIN:

- different **display ways**;
- **area calculation**;
- **TIN re-build** after base layers change;
- **filtering** of base layers, used for TIN creation;
- **accuracy control** of TIN creation;
- **export TIN** to exchange formats.

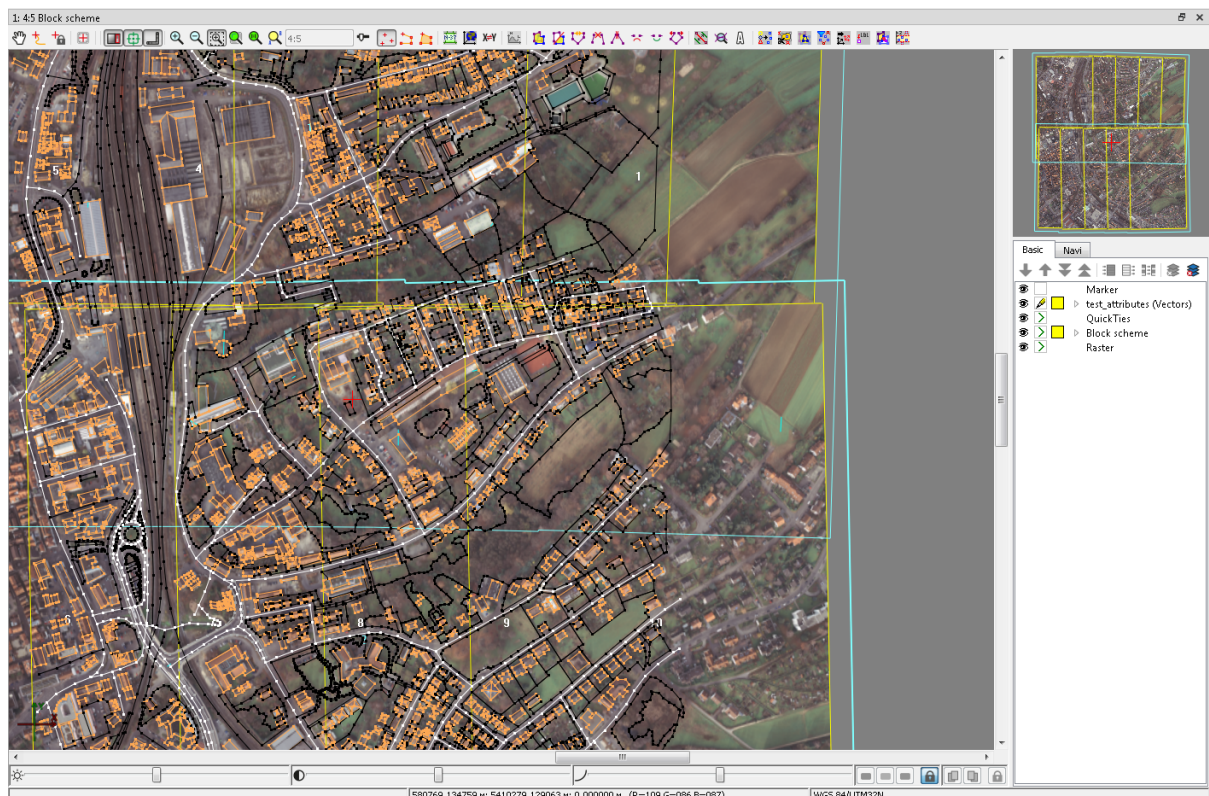


Fig. 28. Source data for TIN creation

5.2.1. TIN layer display settings

For a *TIN* layer could be changed colors of *triangles* and *contours*.

To change colors of objects perform the following:

1. Double-click on layer objects color button () in a *Layer manager*. The **Layer display settings** window opens.

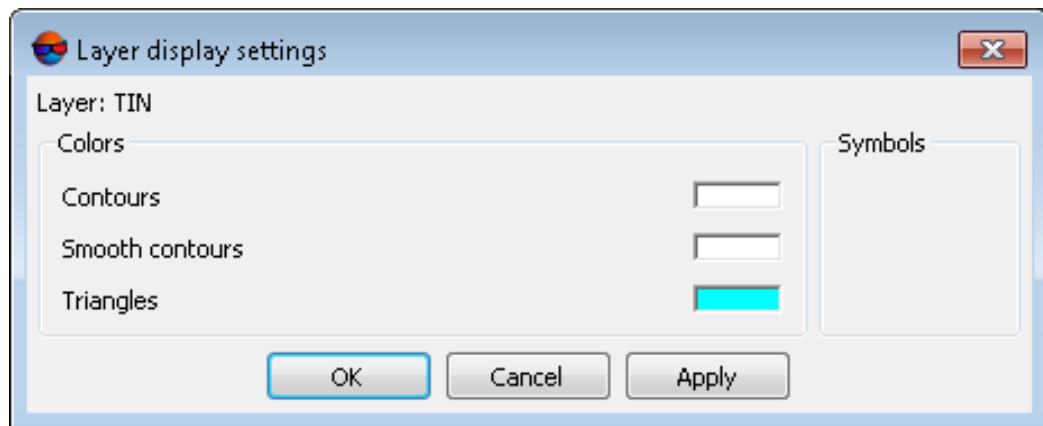


Fig. 29. Layer display settings

2. Double-click on the object's color in the **Colors** section and choose necessary color from standard color palette.
3. Click the **Apply** button to display changes in 2D-window.
4. Click OK to save parameters.

5.3. TIN creation

5.3.1. Creation pre-regions for a block

The system provides possibility to create block pre-regions by images frames or by stereopair frames to allow mutual project processing during vectorization. Pre-regions created by stereopairs also allows to considerably speed up automatic points calculation and generation of dense relief model due to reducing of number of block images overlap areas.



Pre-regions – vector layer which consists of polygons, drawn by images block.

Perform the following actions to create pre-regions:

1. Select **Block > Build pre-regions**. The **Create pre-regions** window opens.

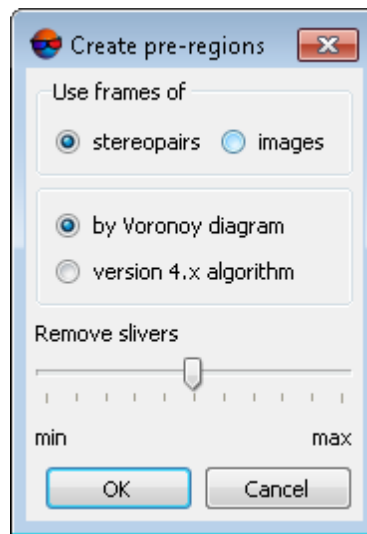


Fig. 30. Parameters of pre-regions creation

2. In the **Use frames of** section select one of the following ways of pre-regions creation:

- **stereopairs** to create pre-regions using overlap areas of two stereopair images;



When using **stereopairs** frames the system fills in polygons attributes automatically as follows: *region_image_code* (with the left image code), *region_image_name* (with the left image name), *region_image_code_2* (with the right image code), *region_image_name_2* (with the right image name).

- **images** to create pre-regions by near-nadir areas of each image.



When using **images** frames the system fills in polygons attributes automatically as follows: *region_image_code*, *region_image_name*.

3. Select a way to create pre-regions using selected frames: Using **Voronoy diagram** or **version 4.x algorithms** earlier system's version algorithm (PHOTOMOD 4.x).

4. [optional] Set the **Remove slivers** slider (at **min** value the pre-regions are built considering narrow strips; at **max** value the pre-regions is created without narrow strips).

5. Click OK to create pre-regions. The system creates a new vector layer with pre-regions.



To load, edit, save and close of pre-regions layer the system provides standard too for work with vector objects (see description in the '*Vector objects editing*' chapter of the [Vectorization](#) User Manual).

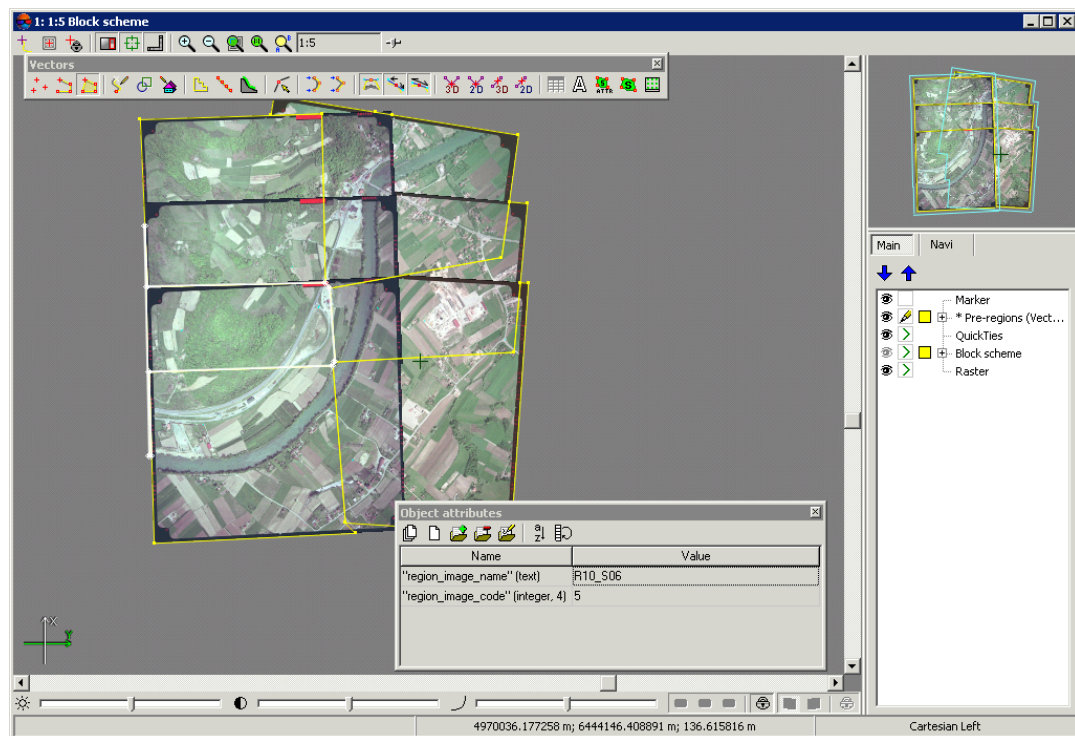


Fig. 31. Pre-regions created using images frames

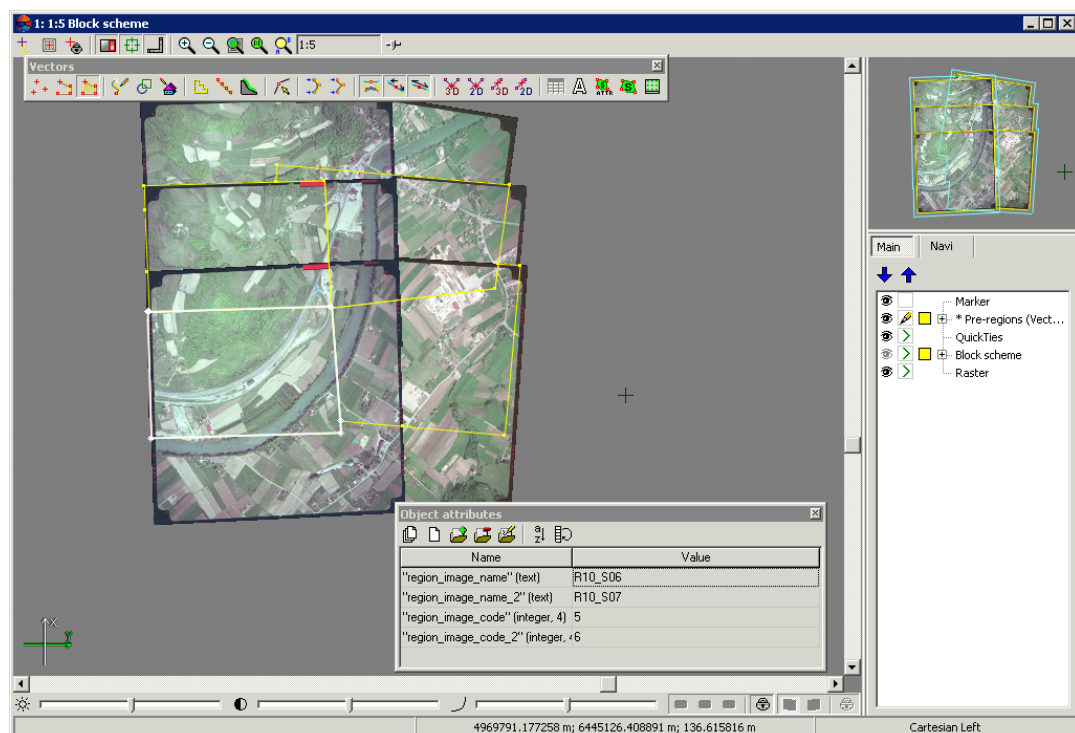


Fig. 32. Pre-regions created using stereopairs frames

5.3.2. Loading base layers for TIN creation

Prior to TIN creation it is necessary to define a set of base layers used to create a TIN.

The system performs TIN creation using layers, containing the following data:

- vector objects (points, polylines and polygons);
- breaklines – polylines/polygons, drawn along lengthy relief features (edges, watershed lines and valley-lines), as well as along roads;
- **set of regular points** – point vector objects, located on relief surface by regular grid or in arbitrary order;
- *triangulation points* – tie, check, GC points, acquired on step of aerial triangulation (see the '[Aerial triangulation](#)' User Manual).

In order to load vector objects perform the following actions:

1. Choose **Vectors** › **Load** or click the  button of the main toolbar. The **Open** window opens.

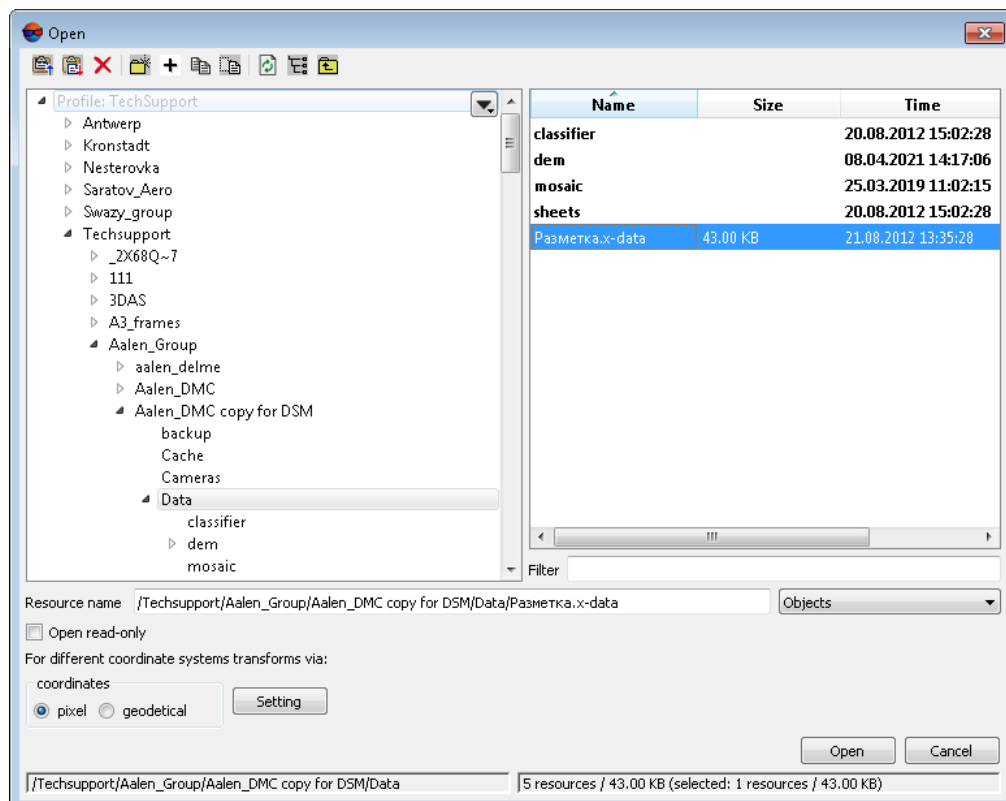


Fig. 33. Vector layer loading

2. Select one or multiple vector files in active profile resources with *.x-data extension.
3. [optional] To disable saving and rewriting of selected file, set the **Open read only** checkbox on.
4. [optional] Set method of objects coordinates recalculation **For different coordinate systems transforms via** – using pixel or geodetic coordinates (see details in the '[General system's parameters](#)' User Manual).
5. Click the **Open** button. If one or multiple vector layers are already loaded to active layer, then during loading of new vector layer the window of loading type is opened.

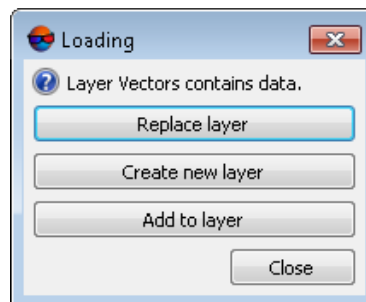


Fig. 34. Vector layer loading

The system provides the following possibilities of new layer loading to existing vector layer:

- **Replace layer** – vector data of active layer is replaced by data from loading layer;
- **Create new layer** – vector data is loaded to a new layer;
- **Add to layer** – during loading vector data is added to objects of active vector layer.



If some point of vector objects layer cannot load, the system shows warning about failed loading of this points. In this case vector layer is loaded without coordinates of this point.

When it is necessary to load two or more vector files at once, the system suggests the following ways of loading:

- **Clear layer and load all files into it** – vector data of active layer is replaced by data from loading layers;
- **Load all into one new layer** – vector data is loaded to a single new vector layer;
- **Create separate layer for each file** – vector data of each file are loaded to separate layers;

- **Load all layers into Vectors without unloading existing data** – during loading vector data is added to objects of active vector layer.

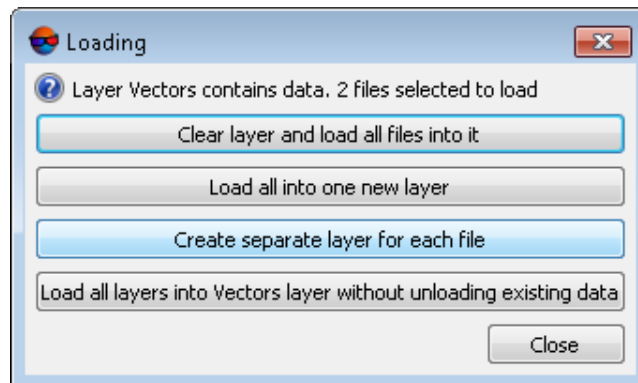


Fig. 35. Vector layer loading

Corresponding *Vector* layers are displayed in *Manager*.

If vector data layer is saved to different coordinate system or in a project with different adjustment results, the system recalculates coordinates automatically and during loading produces information message, that coordinates were recalculated.

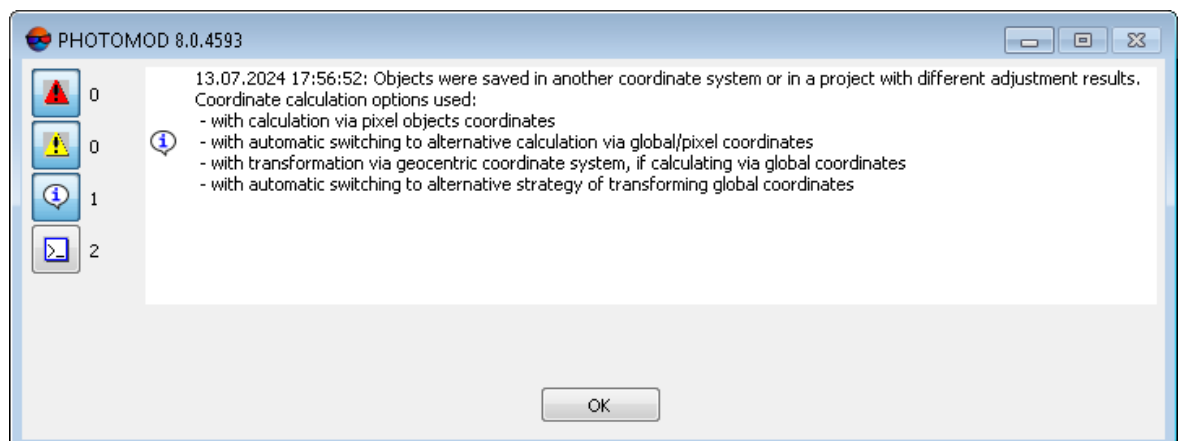


Fig. 36. Coordinates recalculating during vector layer loading

TIN is rebuilt automatically during editing of base layers containing vector objects.



TIN layer is closed during closing of one of base layers.

To use a layer containing triangulation points, perform the following actions:

1. Choose **Vectors › Create layer from triangulation points**. The **Load triangulation points** window opens.

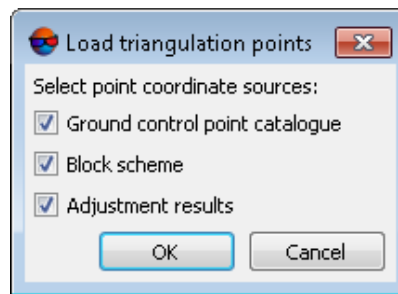


Fig. 37. Load triangulation points to a project

2. [optional] To export triangulation points from GCP list the **Ground control point catalogue** checkbox is set by default.
3. [optional] To export triangulation points from the *Block scheme* layer in 2D-window, the **Block scheme** checkbox is set by default.
4. [optional] To export adjusted points coordinates from GCP catalogue the **Adjustment results** checkbox is set by default.
5. Click OK. The system displays information window containing information about added or skipped points.
6. Click OK. The *Triangulation points (Vectors)* layer is created. Vector point objects (triangulation points) are displayed on block scheme in 2D-window.



Tie points are shown in 2D-window by yellow color; GCP – by red.



There is a possibility to save triangulation points layer to file of *.x-data format to active profile resources.

5.3.3. TIN borders creation

The system provides possibility of TIN borders creation manually or automatically.

To create TIN borders manually, create the border polygons on a separate layer and choose this layer in the list **Use polygons from layer** of the **Create TIN** window.

Perform the following actions to create TIN borders in automatic mode:

1. Choose **Terrain > TIN > Compute border....** The **Compute TIN border** window opens.

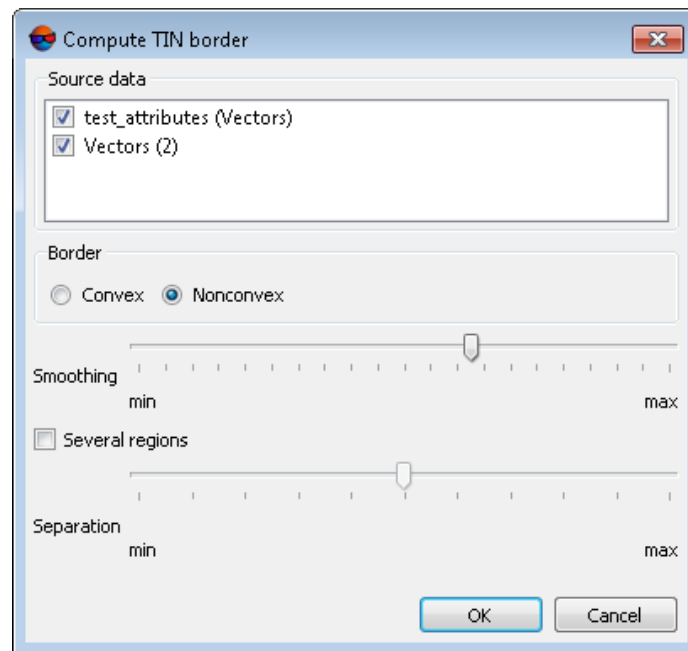


Fig. 38. The 'Compute TIN border' window

2. [optional] In the **Source data** section the checkboxes near all opened layers are set on by default, that are used for TIN borders creation. Set unused checkboxes off, if some layers opened are not used for TIN border creation.
3. [optional] Choose the type of TIN border:

- **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.

4. [optional] In order to split TIN border into areas depending on source data, set the **Several regions** checkbox on and move the **Separation** slider to define maximal distance between initial vector objects, used to create TIN border.
5. Click OK. The system starts to create a TIN border in a new *TIN border* layer.

5.3.4. Building TIN

Perform the following actions to build TIN:

1. Choose the **Terrain > TIN > Build (Ctrl+N, T)** or click the  button of the **TIN** additional toolbar. The **Build TIN** window opens.

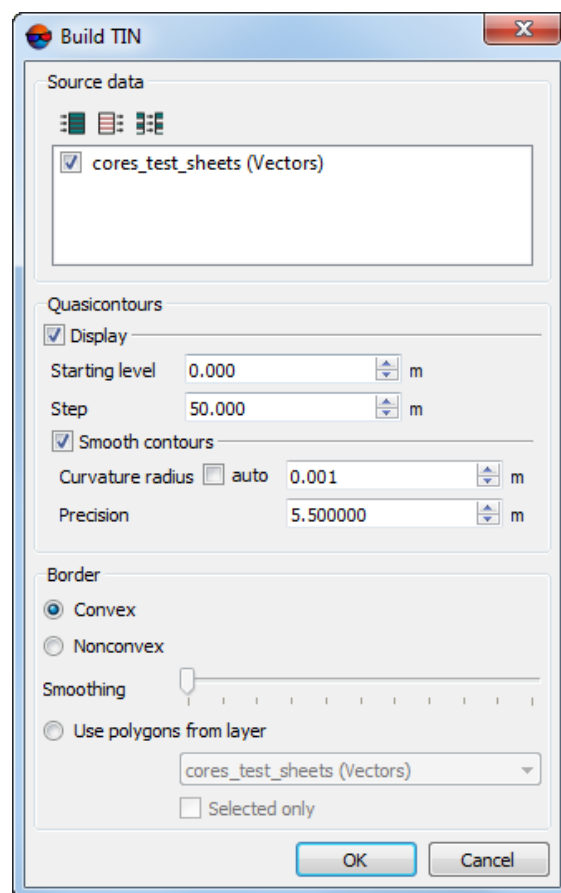





Fig. 39. The Create TIN window

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** field 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.

5.4. TIN display

The system provides possibility to setup TIN display in 2D-window.

In order to make settings choose menu item **Service > Settings** or click the  button of the main toolbar. The **Settings** window opens. It consists of a set of tabs used to setup various parameters.

The **TIN** tab is used to setup parameters of TIN display

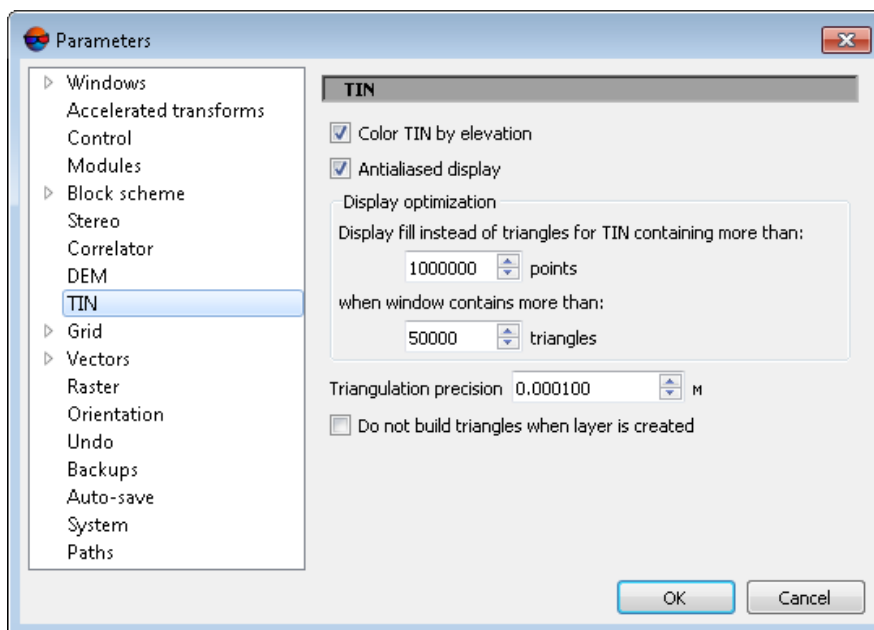


Fig. 40. Settings of TIN display

The **Color TIN by elevation** checkbox is set in the system by default, as a result TIN is displayed as grid of triangles colored by height. Clear the **Color TIN by elevation** checkbox to display all TIN triangles by the same color, defined in the **Manager** panel for TIN layer.

The **Display optimization** section contains the following parameters used to improve performance and to display a TIN as solid fill:

- **Display fill instead of triangles for TIN containing more than** specified number of triangles;
- Display solid fill for TIN **when window contains more than** specified number of triangles.

The **Antialiased display** checkbox is used to smooth TIN and quasi-contours for improving visual perception.



When TIN is displayed as solid fill quasi-contours are not displayed.

The **Triangulation precision** parameter is intended for filtering of errors of TIN creation using vector objects, which contain closely-spaced nodes. A value in the **Triangulation**

precision field is used to specify minimal distance between points, at which the points are considered as different nodes, not the same one.

To save computer memory when editing large datasets set the **Do not build triangles when layer is created**.




If it is necessary to display TIN as triangles when the **Do not build triangles when layer is created** checkbox is on, in the **Display optimization** section input the **Display fill instead of triangles for TIN containing more than** parameter with value that exceeds the previous one.

The system provides a possibility to display TIN as three-dimension relief model, to do this select **Window > 3D window**.

5.5. TIN loading

In order to load a TIN perform the following actions:

1. Select **Terrain > TIN > Load (Ctrl+O, T)** or click the  button of the **TIN** additional toolbar. The **Open** window opens.

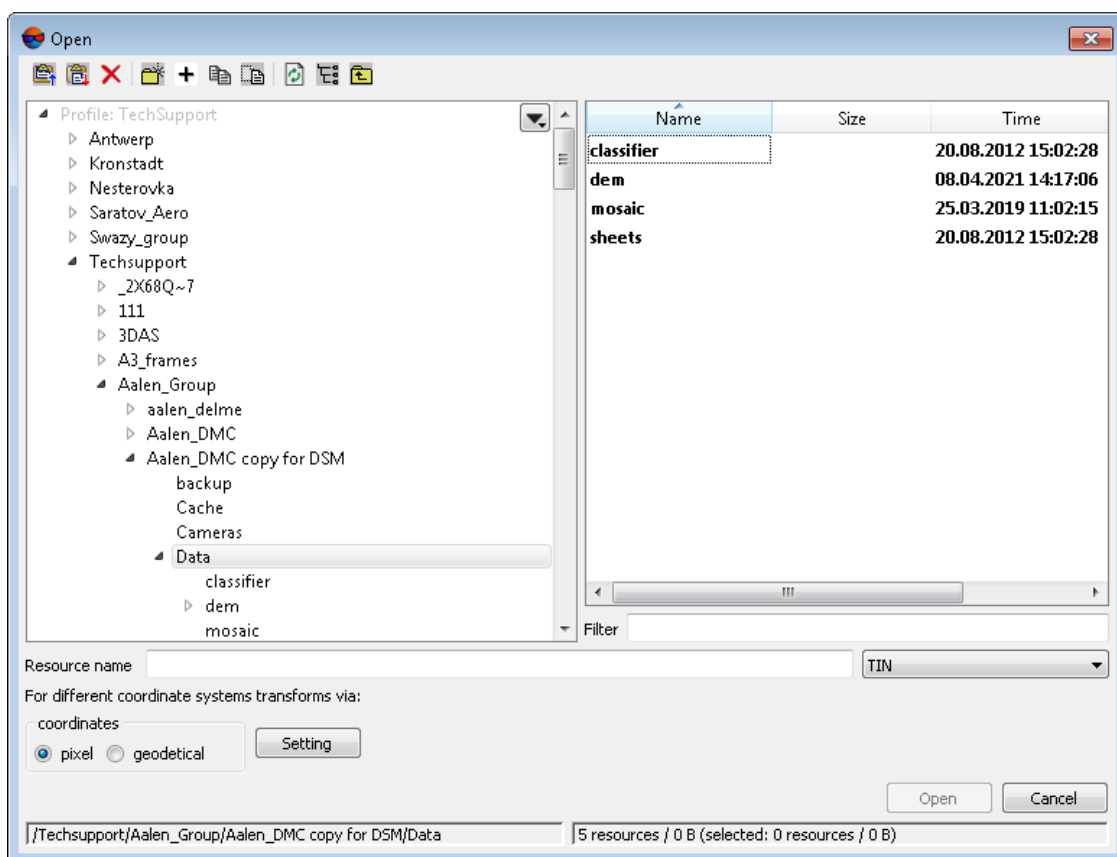


Fig. 41. TIN loading

2. Select one or multiple TIN files in active profile resources with *.x-tin extension.

3. [optional] Set method of objects coordinates recalculation **For different coordinate systems transforms via** – using pixel or geodetic coordinates (see details in the '[General system's parameters](#)' User Manual).
4. Click the **Open** button. After that the TIN layer is loaded along with base vector layers. If one or multiple TIN containing layers are already loaded to active layer, then during loading of a new layer the window of loading type is opened.

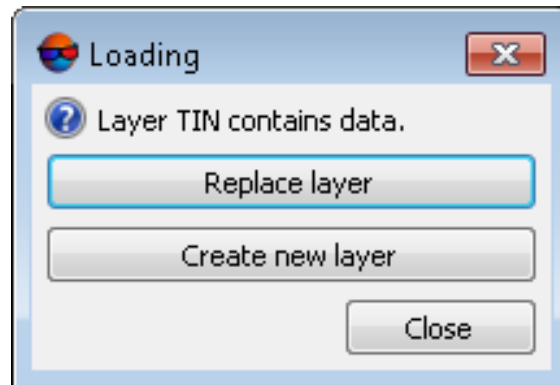


Fig. 42. TIN layer loading

The system provides the following possibilities of new layer loading to existing TIN layer:

- **Replace layer** – vector data of active layer is replaced by data from loading layer;
- **Create new layer** – vector data is loaded to a new layer.

The system provides a possibility of quick access to recently loaded TIN files. To do this use the **Terrain > TIN > Recent** menu item.

5.6. TIN saving

To save and rewrite active TIN layer select **Terrain > TIN > Save** or click the TIN layer name by mouse right button in the *Manager* and choose the **Save** in right-click menu.

Layer containing TIN is saved in the file with x-tin extension. The file contains paths to base layers used to create TIN, and a step of quasi-contours in meters.

To save active TIN layer with new name select **Terrain > TIN > Save as...** or click the TIN layer name by mouse right button in the *Manager* and choose the **Save as...** in right-click menu, specify file name and destination in active profile resources.

During saving a layer containing TIN, base layers used for TIN creation are saved automatically.

During closing of one of base layers, the system suggests to save a layer containing TIN to active profile resources, after that the TIN layer is closed automatically.

5.7. TIN restoring

The system provides possibility to restore TIN from contours. During TIN restoring the additional vertices between the contours should be created.

This feature set can be useful in case of processing input data that are contours with an irregular interval, unsuitable for solving current problems. To restore the terrain from the input contours, in order to further create contours with the desired step, perform the following:

1. **Restore** TIN by contours;
2. **Build** DEM by TIN
3. **Build** contours by DEM



Using contours as input data for TIN creation, with correct user settings, results in the best quality of terrain display with a triangulation irregular network (compared to other types of input vector objects).

The quality of TIN building, in turn, affects the correct terrain display by DEM and contours that will be built later using the abovementioned TIN as source data.

5.7.1. TIN restoring from contour lines

Perform the following actions to restore TIN:

1. Choose the **TIN › Restore....** The **Restore TIN** window opens.

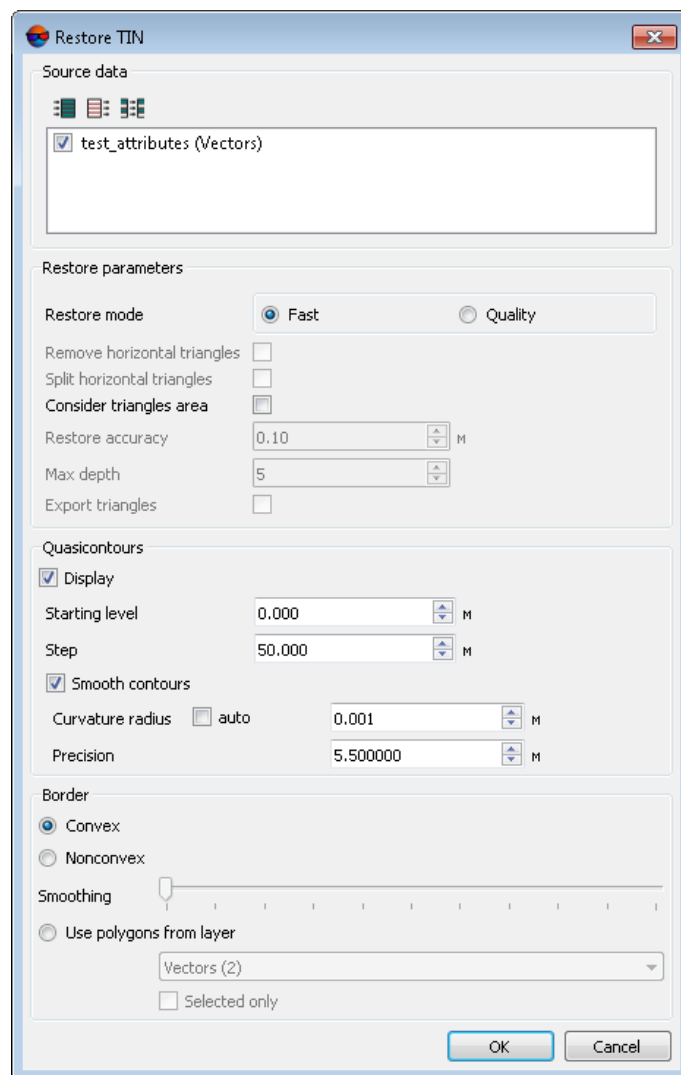





Fig. 43. The Restore TIN window

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



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. Select the **Restore mode** in the **Restore parameters** section:
 - **Fast**;
 - **Quality**;
 - [optional] set the restore accuracy and triangles splitting parameters;

4. [optional] In the **Quasicontours** section the **Display** checkbox on and in the **Starting level** field specify minimal Z level (Zmin), from which to start quasi-contours creation.
5. [optional] Specify the **Interval** of quasi-contours creation in meters.
6. [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.
7. 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.
8. Click OK. The TIN is created in the new *TIN* layer. The additional vertices, created during TIN restoration, are situated in the new *Ridges* vector layer.

5.8. TIN info

5.8.1. Brief TIN info

To get brief information about TIN layer, click the TIN layer name by mouse right button in the *Manager* and choose the **Information** in right-click menu. After that the system displays information window which includes the following information about the layer:

- layer name;
- full path to layer file;
- presence of data and changes in the layer;
- possibility to change the layer;
- coordinates of TIN border nodes, including Z-coordinate;
- number of nodes in base layers;
- layer coordinate system.

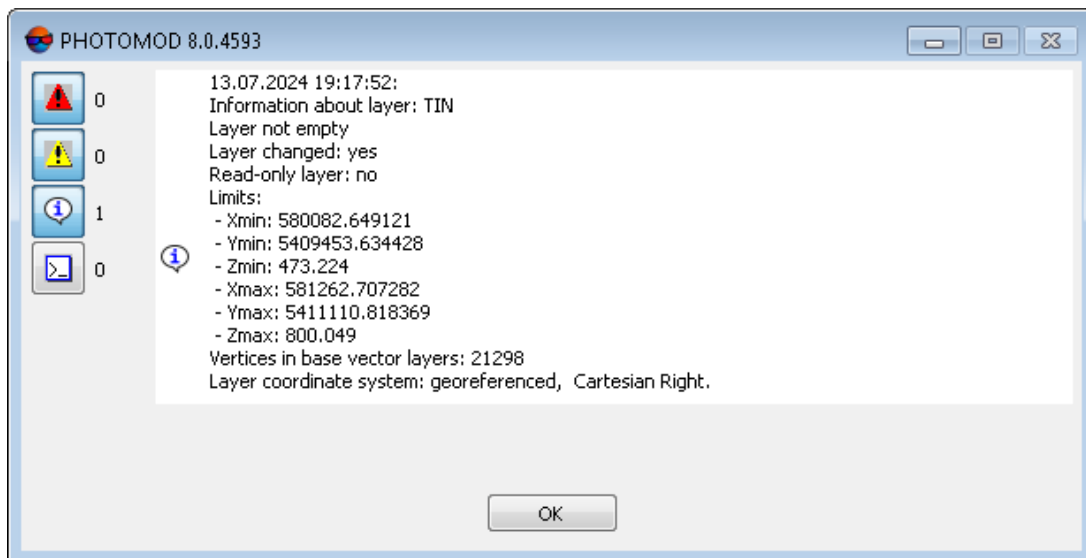


Fig. 44. TIN layer info

5.8.2. General TIN info

In order to obtain detailed information about TIN layer, choose **Terrain > TIN > Statistics**. The **TIN – statistics** window opens.

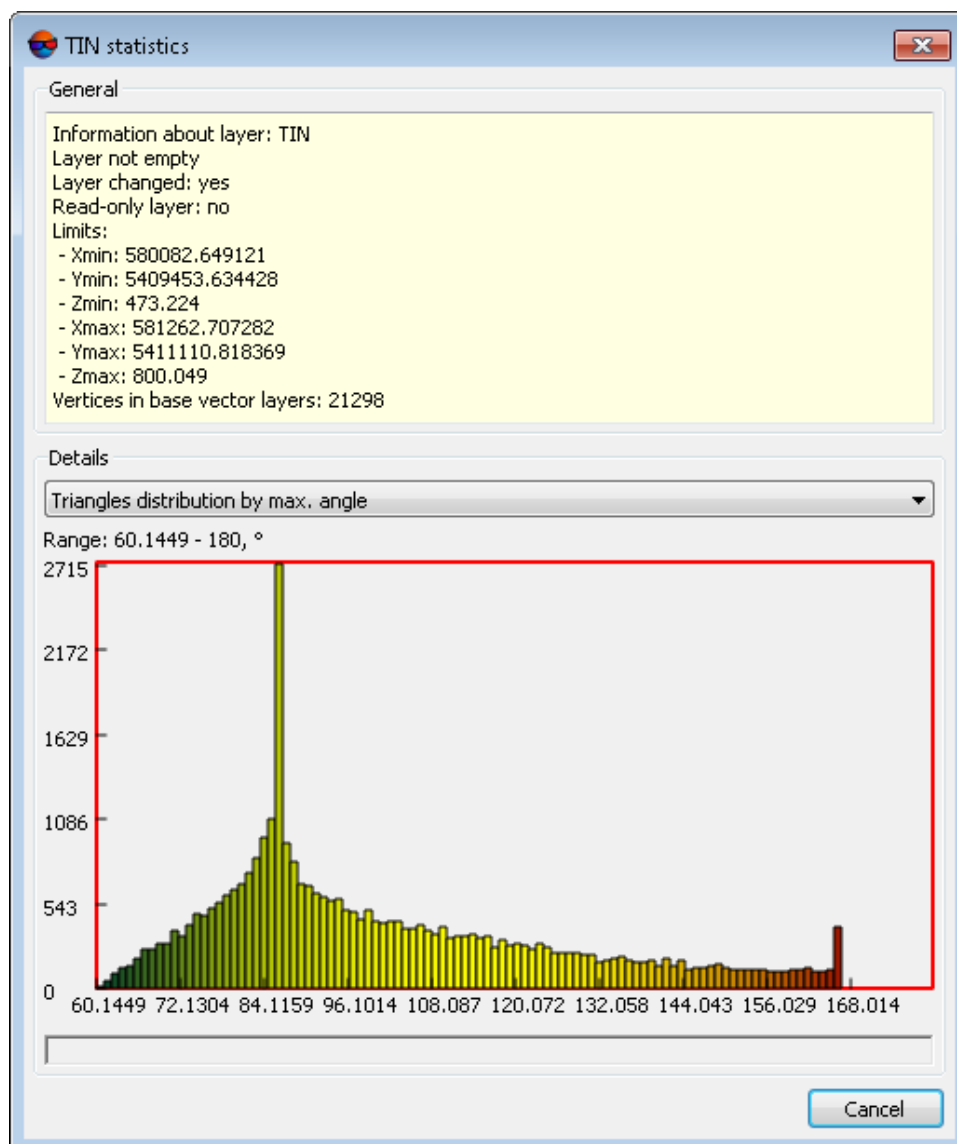


Fig. 45. The TIN statistics window

The **General** section includes [brief information](#) about TIN layer.

The **Details** section contains information about distribution range of the following data on a layer:



It is recommended to use this information to search for blunders on TIN layer.

- **Triangles distribution on area** (in square meters);
- **Nodes distribution along Z axis** (in meters);
- **Triangles distribution by min. angle** (in degrees);

- **Triangles distribution by max. angle** (in degrees).

Below is a table of the data distribution, which contains the following lines:

- **Min** and **Max** – minimal and maximal data range;
- (for example, 28588.2 - 31764.6) – data distribution in percentage.

5.8.3. TIN area info

The system provides possibility to calculate TIN surface area.

To calculate TIN surface area within polygon, select the polygon on base layer or create a new polygon and select **Terrain > TIN > Compute surface area inside polygon**. After that the system opens information window with the following data:

- information about value of TIN area within selected polygon;
- information about value of selected polygon area.

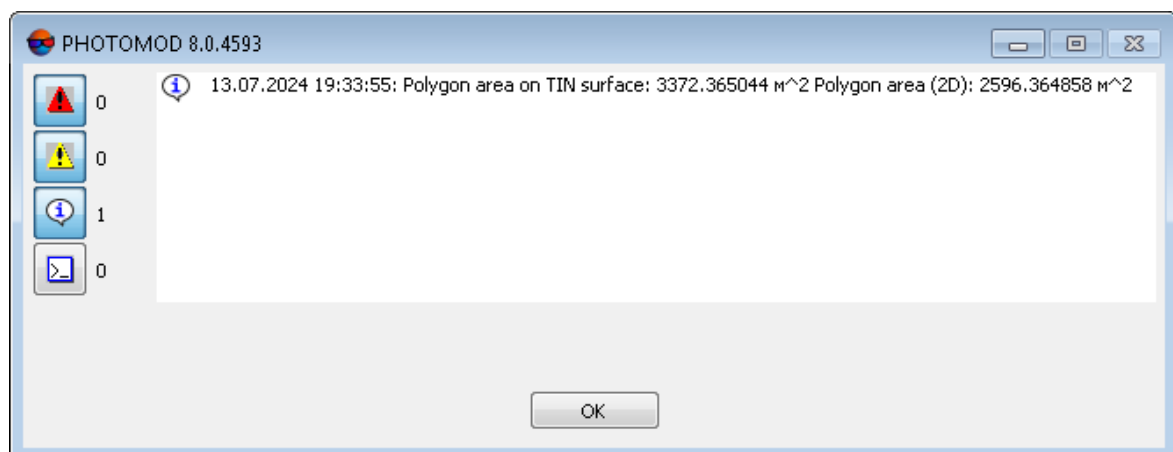


Fig. 46. The area of a polygon on TIN surface

In order to calculate area of TIN projection on a plane and area of TIN 3D surface, select **Terrain > TIN > Compute surface area**. After that the system displays information window with values of TIN areas:

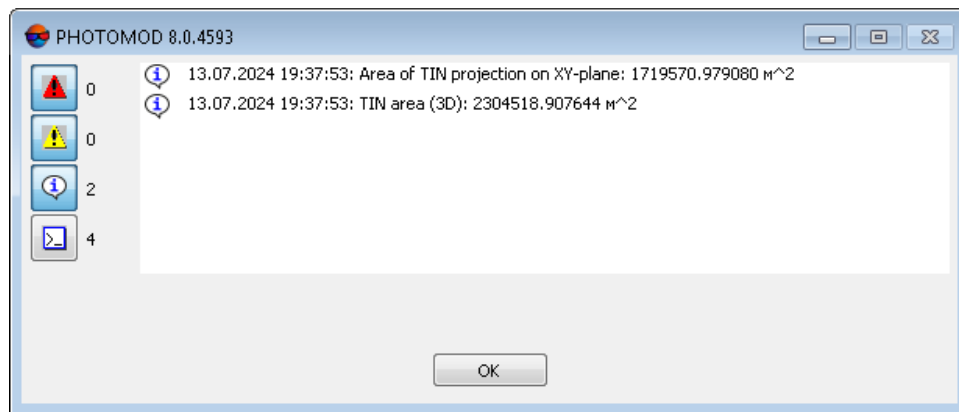


Fig. 47. TIN surface area

5.9. TIN re-building

After editing of base layer objects (see the '[Vectorization](#)' User Manual) the system allows to perform TIN re-build operation.



TIN layer is not editable. To change TIN layer it is necessary to edit base layers used for TIN creation.

In order to re-build TIN considering vector objects of new vector layer, select **Vectors › Load**, select a file and click the **Open** button. The **Loading** window opens.

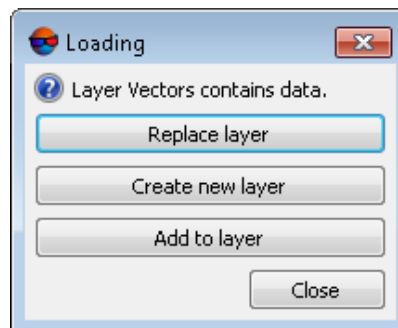



Fig. 48. Vector layer loading

The system provides the following possibilities of new layer loading to existing vector layer:

- **Replace layer** or **Add to layer** (without unloading existing data) – allows to load vector objects to a base layer. After that the TIN is rebuilt automatically;
- **Create new layer** – allows to load vector objects to a new layer. [Create a TIN](#) using new layer as a base one.

To re-build TIN considering changes made in vector objects located in base layers, perform the following actions:

1. Choose **Terrain > TIN > Rebuild** or click the  button of the **TIN** additional toolbar. The **TIN border smoothing** window opens.

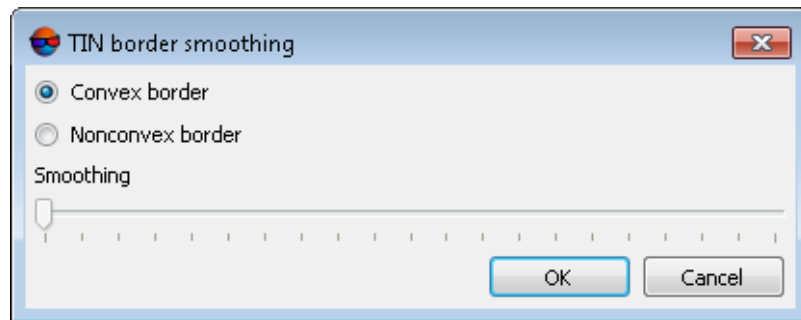


Fig. 49. TIN border smoothing

2. Set the following parameters:

- **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.

3. Click OK to perform TIN re-build according to changed data.

5.10. Insert objects into TIN

The system provides possibility to insert objects into existed TIN layer.

It is used to display vertical objects (e.g. walls of buildings) in the TIN layer.

Embedded objects are to be located in the loaded vector layers that are not basic ones for the TIN layer being edited.

To insert vector objects into TIN layer is used the **Terrain › TIN › Insert objects** menu item.

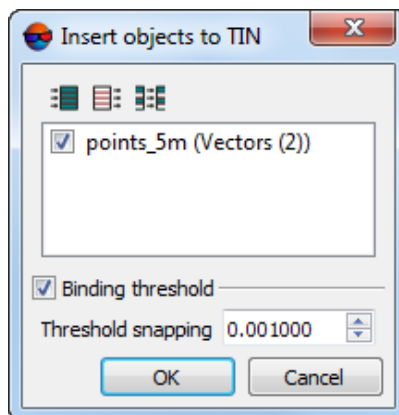





Fig. 50. Insert objects to TIN

To choose vector file are used the following buttons:

-  – allows to select all files;
-  – allows to unselect all files;
-  – allows to invert files selection.

New layer creates when process complete. If the **Binding threshold** checkbox is set on, the new layer with inserted objects and TIN is created. Otherwise, only vector objects are created in new layer.

Threshold snapping allows to set the value in meters. If the value less than threshold, objects are not insert.

5.11. Accuracy control of TIN creation

5.11.1. Check against adjustment points

If triangulation points layer was not used during TIN creating, the system provides possibility to perform accuracy control of TIN creation by triangulation points, acquired on aerial triangulation step. To do this use the **Terrain › TIN › Check against adjustment points** menu item.



Triangulation points must have the adjusted coordinates (**X**, **Y**, **Z**).



It is not necessary to create points vector layer.

The **Accuracy control** window includes a table of all points (GCP/check/tie), acquired on an aerial triangulation step.

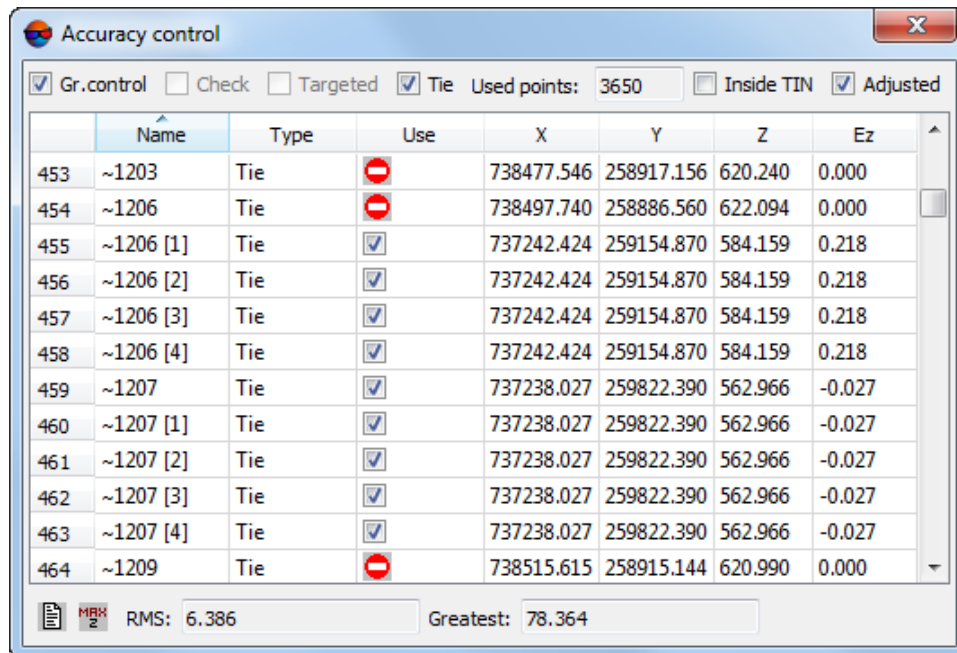


Fig. 51. Accuracy control using triangulation points

Checkboxes in the toolbar at the top of the window are used to customize the display of triangulation points in the list according to their types and properties (see the “Point properties” chapter in “[Aerial triangulation](#)” User Manual):

- **Gr.control** – set the appropriate checkbox to display points of this type in the table;;
- **Check** – set the appropriate checkbox to display points of this type in the table;
- **Tie** – set the appropriate checkbox to display points of this type in the table;
- **Targeted** – set the appropriate checkbox to display **targeted** points in the table.



Only those types of triangulation points that present in this project are available for selection.

To use the **adjusted** points to check the TIN accuracy – set the appropriate checkbox.




Clear this checkbox to perform accuracy control by triangulation points disregarding adjustment.


To display in the table only the points **inside TIN**, set the appropriate checkbox.

The **Used points** field is used to display number of points used for calculation of DEM deviation by Z-coordinate (in the **Use** column the ☒ is set).


The table of triangulation points contains the following columns:

- Name – point number;
- Type – point type: GCP, check or tie point;
- Use – data about point use for calculating of DEM deviation by Z-coordinate:
 - ☒ – the triangulation point was used;
 - ☐ – the triangulation point was not used;
 -  – the triangulation point cannot be used, since it was not used for adjustment.
- X, Y, Z – adjusted coordinates of triangulation points;
- Ez – value of DEM deviation from the triangulation point by Z-coordinate.

Status panel is used to display values of root mean square error (RMS) and a value of maximal error of DEM deviation by Z-coordinate (Max).

The  button allows to display in the list a point with maximal error of deviation from triangulation points by Z-coordinate.

The  button allows to display detailed report on accuracy control.

 The **report** is generated according to the data currently displayed in the triangulation points table.

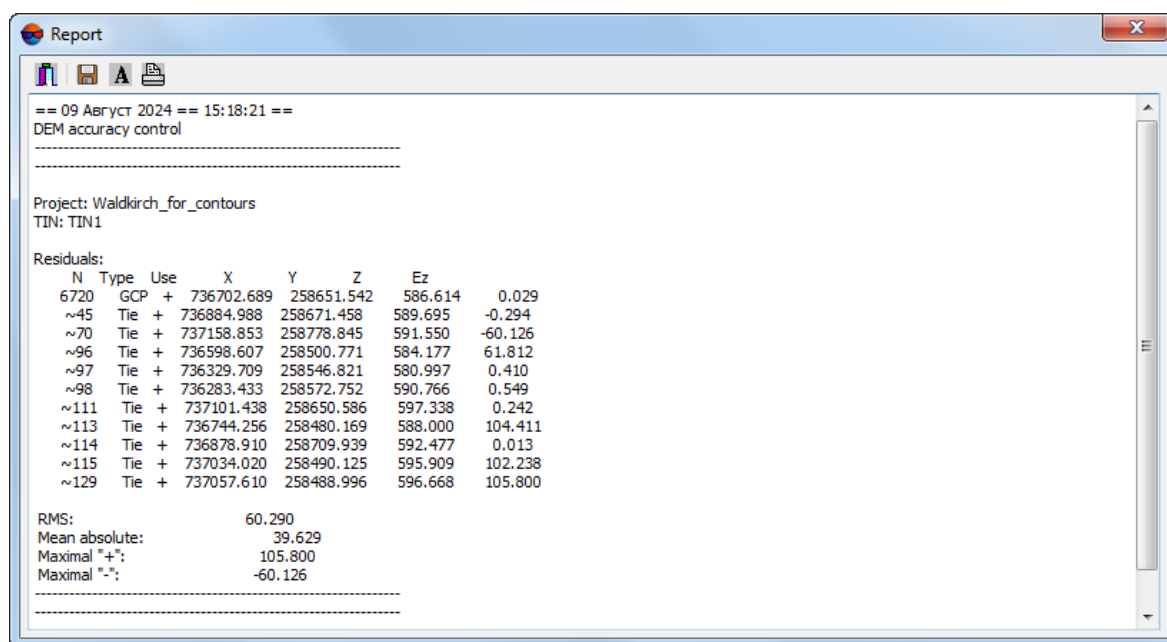






Fig. 52. Report on accuracy control using triangulation points

The top panel of the **Report** window contains the following buttons:

-  – allows to close report window;
-  – allows to save a report to file system;
-  – allows to change font, style, size, and color of report text;
-  – allows to print a report.

5.11.2. Check TIN against vector objects

The system allows to control TIN creation accuracy by vector objects, which were not used during TIN creation Control operation provides values of TIN deviation from vector objects vertices by Z-coordinate.

To control TIN creation against vector objects perform the following actions:

1. Load the appropriate vector layer and make it editable;
2. Select **Terrain > TIN > Check TIN against vector objects**. The **Parameters of check TIN by vectors** window opens.

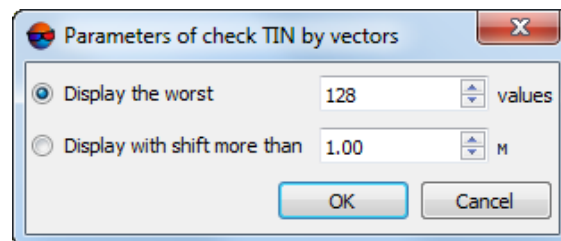
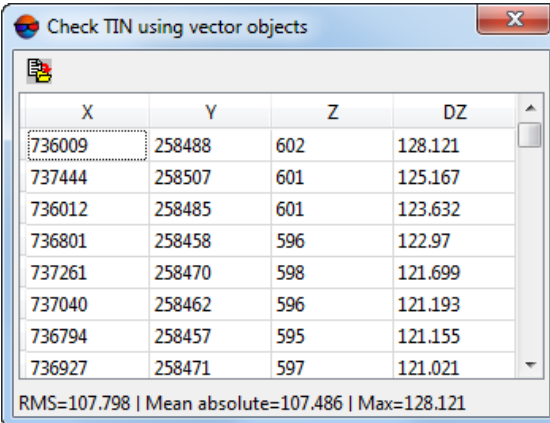


Fig. 53. Parameters of check TIN by vectors

3. Select display parameters of points list:
 - **Display the worst..values** – allows to display a list of vector objects vertices with worst deviations from TIN by Z-coordinate;
 - **Display the the shift more than** – allows to display a list of vector objects vertices with deviation from TIN by Z-coordinate, that exceeds specified.
4. Click OK. The system starts operation of comparing of TIN deviation values from vector objects vertices by Z-coordinate. After that the **Check TIN using vector objects** window opens.



X	Y	Z	DZ
736009	258488	602	128.121
737444	258507	601	125.167
736012	258485	601	123.632
736801	258458	596	122.97
737261	258470	598	121.699
737040	258462	596	121.193
736794	258457	595	121.155
736927	258471	597	121.021

RMS=107.798 | Mean absolute=107.486 | Max=128.121

Fig. 54. Results of control check of TIN against vector objects

In order to export points list to file with *.csv extension, click the  button.

5. Select a point in the list by mouse click. After that marker is placed to the crossing point and is displayed in 2D-window.

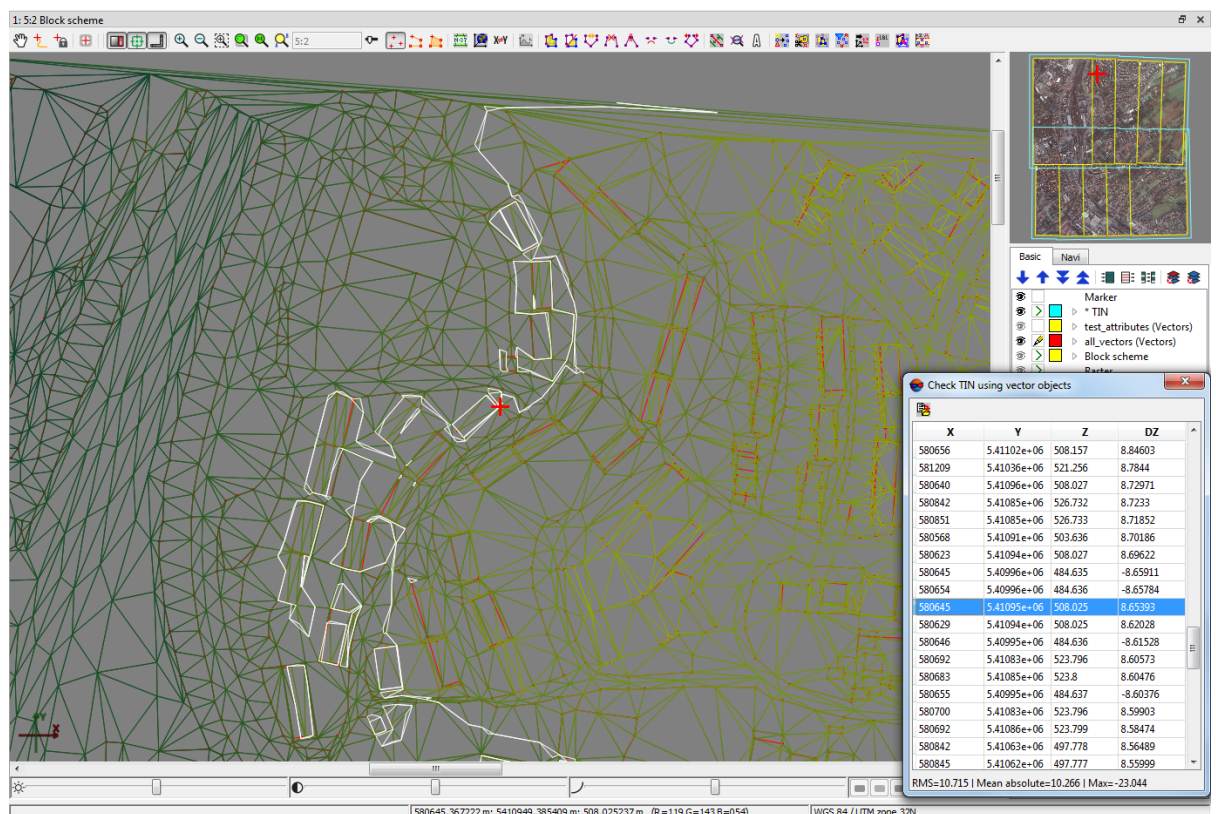


Fig. 55. Check TIN against vector objects

5.11.3. Verifying of TIN topology

The system provides possibility to verify TIN topology.

The *topology* concept is one of the key terms used during quality assessment of created maps. Throughout this documentation, the *topology* is referred to as set of functions and operations, that define mutual location of vector objects and are used to create topologically correct vector data.



Topology (from Greek *topos* - place) – a branch of mathematics that studies the phenomenon of continuity in the most general form, in particular, the properties of space, which remain unchanged under continuous deformations, for example, the connectivity.

Check of TIN topology means searching for closely placed triangles nodes of TIN and their correction (correction of their corresponding points or vertexes of polylines/polygons of base vector layers). Topology correction includes deleting or moving of closest TIN nodes.

In order to start TIN topology check, perform the following actions:

1. Select **Terrain > TIN > Verify topology...**. The **Parameters** window opens.

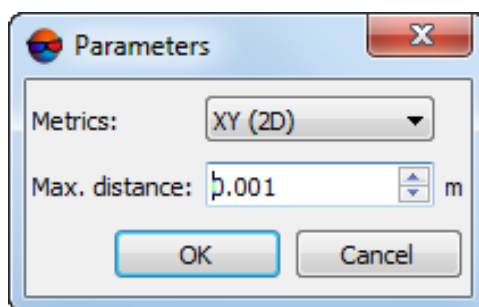


Fig. 56. Parameters of topology check

2. In the **Metrics** list specify whether 2D or 3D space is used to search for topology errors.
3. In the **Max. distance** field define a distance between nodes, below which the nodes are considered closest.
4. Click OK. The system starts operation of search for closest TIN nodes. When the search is completed the system opens the **Errors correction** window.

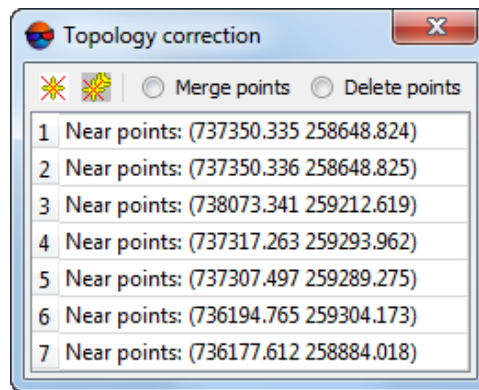




Fig. 57. Correction of topology errors

5. Select a point in the list by mouse click. After that marker is placed to the point and is displayed in 2D-window.
6. Select a way of topology correction: **Combine points** or **Delete points**.
7. To correct closest TIN nodes perform one of the following actions:
 - 'link' nodes using snapping (see the '[Vectorization](#)' User Manual);
 - to correct topology for all points in the list click the  button;
 - to correct topology only for selected points in the list click the  button.

After correction of TIN closest nodes topology the erroneous nodes are combined into a single node with average coordinates value, or are deleted.

5.12. Export TIN

5.12.1. Export TIN to DXF

The system provides possibility to export of TIN layer to DXF format for its further use in third-party software.

In order to export TIN to DXF format perform the following actions:

1. Select **Terrain** › **TIN** › **Export** › **DXF...**. The **Export to DXF** window opens.
2. Choose a folder to place a file to file system.
3. Click the **Save** button. The **Export to DXF** window opens.

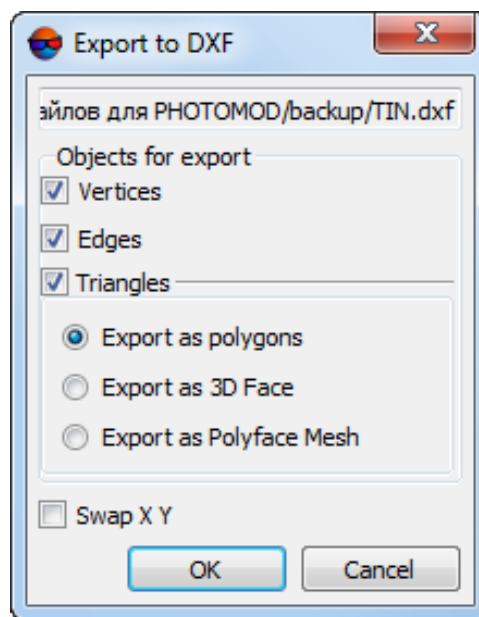


Fig. 58. Parameters of export to DXF format

4. [optional] In the **Objects for export** section to export vertices, edges and triangles of TIN the **Vertices**, **Edges**, **Triangles** checkboxes are set on by default. Clear the checkboxes to export just particular kind of objects.
5. Set the following parameters:
 - **Export as polygons** – allows to export objects as polygons;
 - **Export as 3D Face** – allows to export top face of an object;
 - **Export as Polyface Mesh** – allows to export object as polyhedral object.
6. [optional] In order to swap coordinate system (left/right) during TIN export, set the **Swap X Y** checkbox.



If the **Swap X Y** checkbox is off, the initial data will be imported in the right coordinate system. Otherwise, in the left coordinate system.

7. Click OK to complete the export operation.

5.12.2. Export TIN to CSV

The system provides possibility of TIN export to CSV format. 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.

In order to export TIN to CSV format perform the following actions:

1. Select **Terrain > TIN > Export > CSV....** The **Export to CSV** window opens.
2. Choose a folder to place a file to file system.
3. Click the **Save** button. The **Export to CSV** window opens.

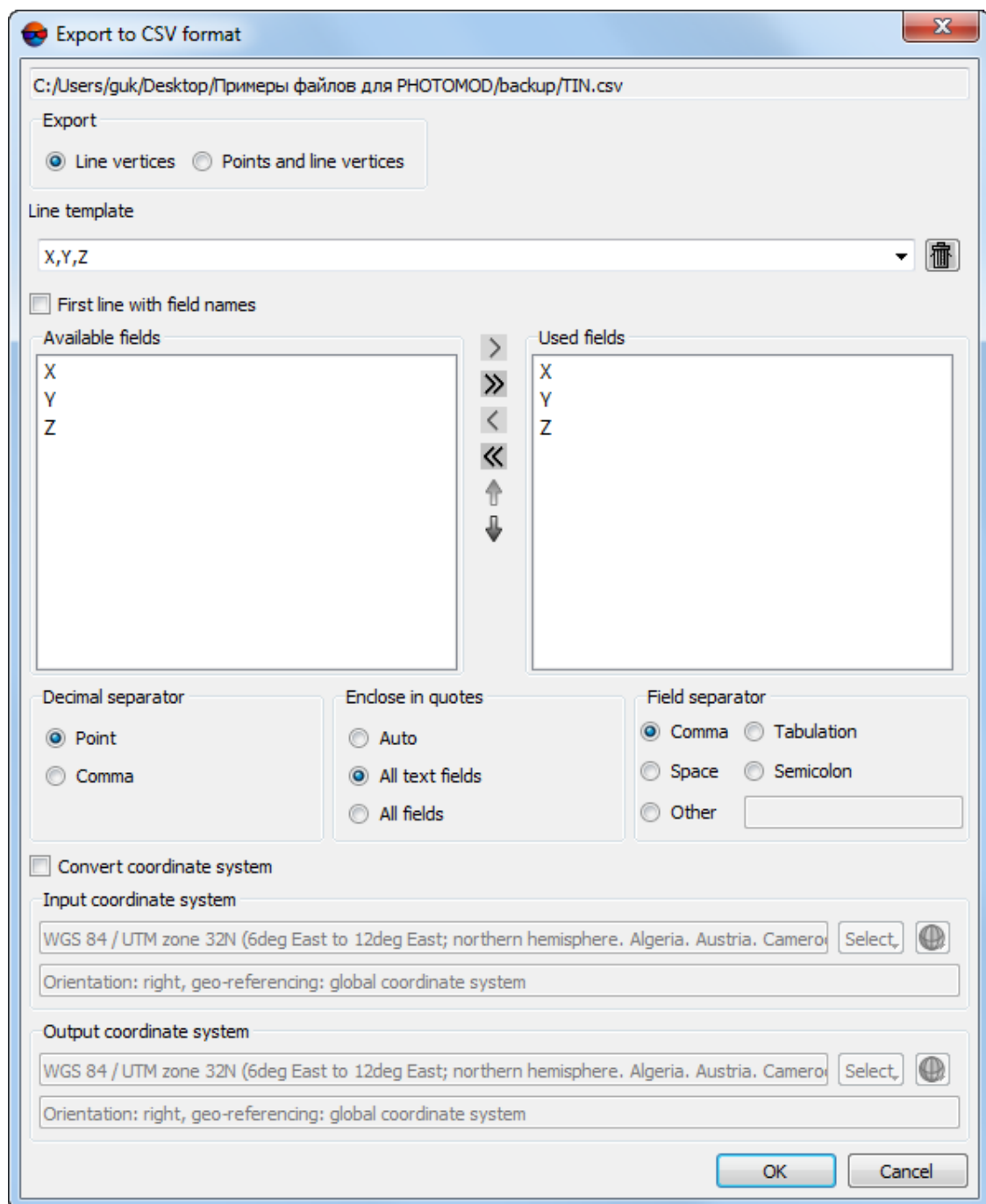









Fig. 59. Parameters of export to CSV format

4. In the **Export** section select what kind of objects it is necessary to export: **Line vertices** or **Points and line vertices**.
5. The **Line template** field displays the list of fields, contained in each line of exported CSV file: To clear the field, click the **Clear the list**  button.
6. [optional] In order to export a line from the **Line template** field, set the **First line with field names** checkbox on.
7. [optional] In order to change number of decimal places in coordinates to be exported, set the **Maximum decimal places:** checkbox on and input needed value.
8. In the **Available fields** list select required field name and click the **Add selected field**  button or click the **Add all fields**  button, to transfer all field names. After that all or selected field names are moved to the **Used fields** list.



To cancel field name selection click the  button or click the  button to cancel moving all field names. To move selected field to the list bottom, click the  button, and to move selected field to the list top click the  button.

9. In the **Decimal separator** section select point or comma to be used to separate coordinates.
10. [optional] In order to limit required parts of exported list of coordinates by quotes, in the **Enclose in quotes** section choose one of the following options:
 - **Auto** – allows to limit by quotes the fields, which are located in the file to be exported;
 - **All text fields** – allows to limit by quotes only the fields, which contain text information;
 - **All fields** – allows to limit by quotes each field, which is located in the exported file.
11. In the **Field separator** section choose, what is used to separate fields: comma, space, tab, semicolon or other delimiters.



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.

12. [optional] To define standard parameters set the **Transform reference system** checkbox on (see the chapter “Transform reference system of objects” in the [‘Vectorization’ User Manual](#));
13. Click OK to complete the export operation.

5.13. Import TIN

5.13.1. Import from DXF



The DXF format, that is used in the *AutoCAD R15* and later versions is not supported. In this case to perform data exchange it is necessary in *AutoCAD program* to convert DXF file to R14 format.

The system provides possibility of vector objects import from DXF format (see the “[Vectorization](#)” User Manual). This is exchange format with *.dxf extension, that is used in the *AutoCAD programs*.

In case when 3D face objects are to be imported from the DXF format, for the user’s convenience the system provides for immediate building the TIN layer on the base of imported 3D face objects.



A user is able to import 3D face objects as a vector layer (see the “[Vectorization](#)” User Manual) and further to [build a TIN](#) on its base.

The settings for both ways of data import from the DXF format are the same, because in any case, during import, a vector layer will be created at first, which will either be used for automatic TIN construction, or processed somehow by the user, depending on specific needs.

Note, that 3D face objects can be represented both by triangles and polygons. Their structure will be completely maintained when importing 3D face objects as a vector layer. When building a TIN (in every way) polygons will be split into triangles anyway.

In order to import TIN from the DXF format perform the following actions:

1. Choose **DEM › TIN › Import › DXF**. The **Import from DXF format** window opens:

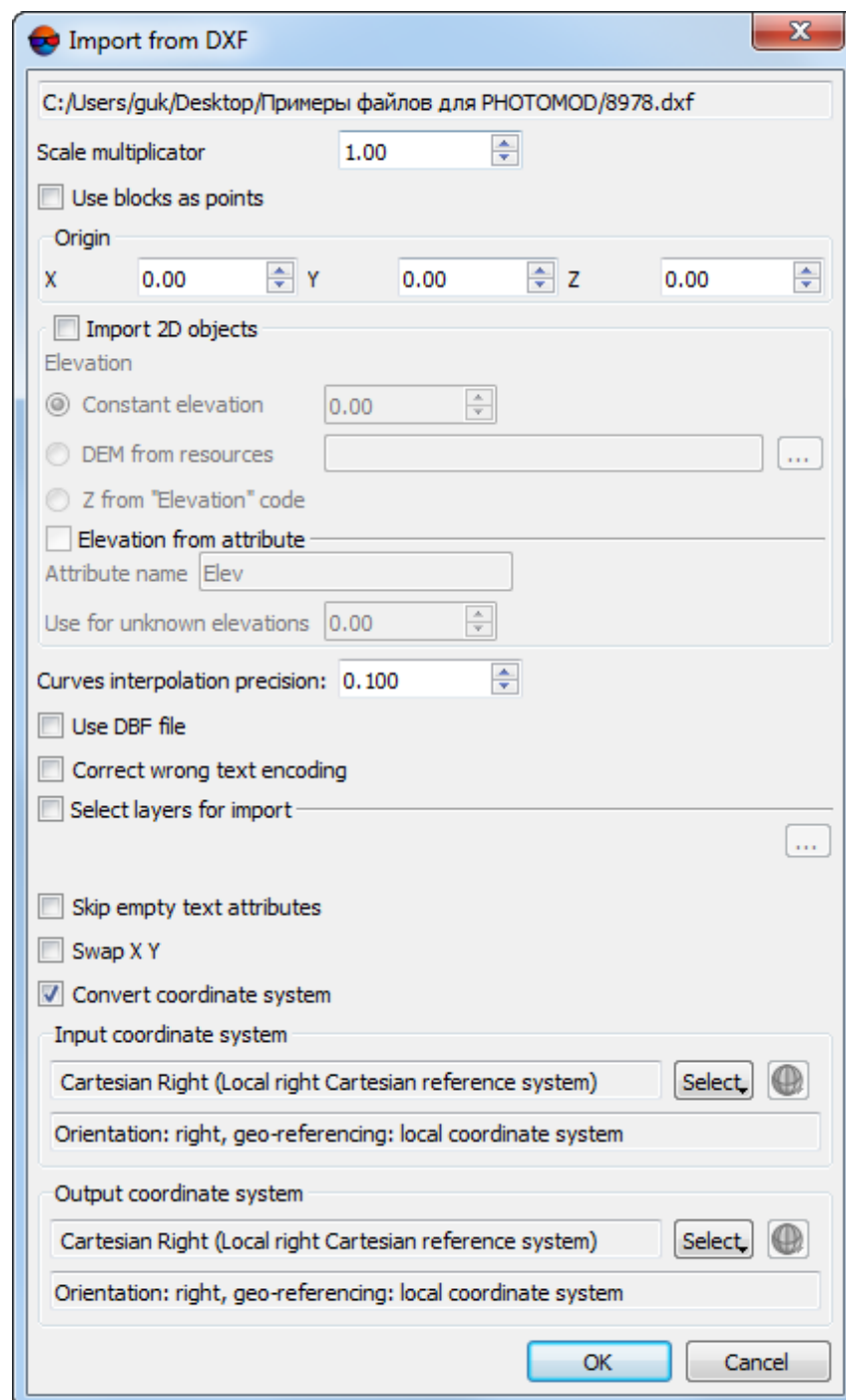



Fig. 60. Import from DXF

2. [optional] To multiply all coordinates of objects vertices during import specify the coefficient in the **Scale** field;
3. [optional] To import origin of *AutoCAD program* objects blocks as point objects, set the **Use blocks as points**;

4. In the **Point of origin** section specify origin of coordinate system;
5. [optional] To convert 2D objects from DXF file into 3D objects, set the **Import 2D objects** checkbox on. In the **Z for 2D objects** section define one of the calculation ways of Z coordinates of objects vertices:

- **Constant elevation** – allows to assign the same value of Z coordinate to all objects;
- **DEM from resources** – allows to select a resource with DEM containing Z values. Click the  button to choose file in active profile resources;
- **Z from “Elevation” code** – allows to import Z coordinate from “Elevation” code.

The **Elevation from attribute** checkbox allows to import Z coordinate from attribute, which name is specified in the **attribute name** field, its default value is *Elev*.

The **Use for unknown elevation** field allows to fill the empty elevations with a predetermined value.

6. [optional] To define import accuracy of objects, containing curve lines, specify the **Curves interpolation precision** parameter value.



After that the number of curve vertices may change.

7. [optional] To prevent creating text attributes with an empty 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.

8. [optional] To define additional parameters set the **Use DBF file** (see the “[Vectorization](#)” User Manual), **Swap X Y** and **Transform reference system** checkboxes on (see the “[Vectorization](#)” User Manual).



If the last two checkboxes are set on, then when importing first XY-coordinates of points are swapped, then the system recalculates the coordinates.

9. [optional] To correct wrong import of text objects set the **Correct wrong text encoding** checkbox on.
10. [optional] To import a particular number of layers set the **Select layers for import** checkbox on and click the  button. The **Layers table** window is opened, where the list of layers to be imported is displayed.

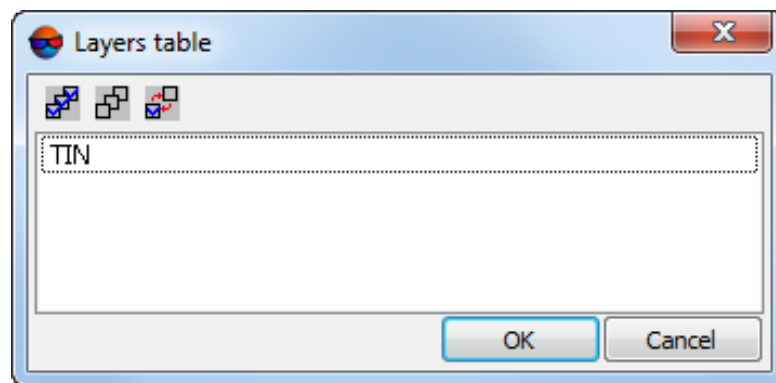



Fig. 61. Table of layers to be imported

Select layers in the table using one of the following ways:

- to import all layers, click the  button and click OK;
- to import not all layers, select them and click OK.



to invert layers selecting allows the  button, to unselect all layers – the  button.

11. Click OK to import.

5.14. TIN filtering

5.14.1. TIN simplifying

The system allows to filter TIN triangles nodes using filtering of point vector objects in base layers, used for TIN creation.



TIN filtering is performed considering breaklines.

To perform points simplifying perform the following actions:

1. **Create** or **load** TIN.
2. Select **Terrain > TIN > Simplify**. The **Simplify TIN** window opens.

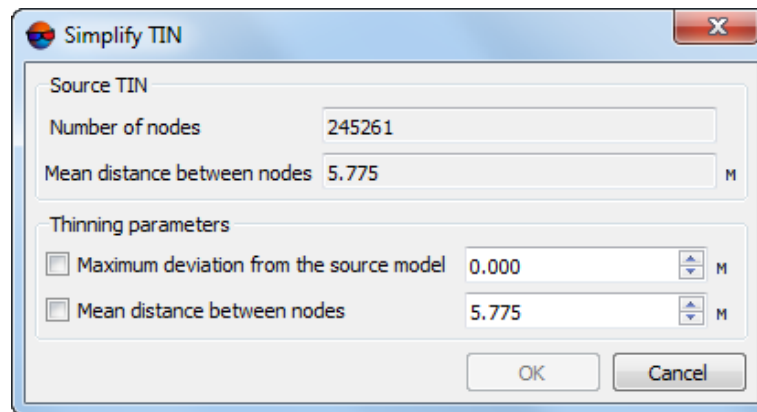


Fig. 62. Parameters of TIN simplifying

The window displays a statistics about source TIN: total **Number of nodes** in network and **Mean distance between nodes** (average length of TIN edges) in meters.

3. Specify **Thinning parameters**:

- [optional] set the **Maximum deviation from the source model** checkbox and specify maximum deviation between source and thinned models by Z in meters;



The maximum deviation is calculated between the nodes of the source TIN and the surface of the thinned TIN.



- [optional] set the **Mean distance between nodes** checkbox and specify distance between nodes after thinning in meters.

4. Click OK. When the filtering operation is completed the system displays information message about number of deleted nodes.



Deleting of TIN nodes means removing of vector points from base layers. The filter works with all base layers as with a single editable layer.



To cancel thinning results the **Undo log** is used, as well as the  and  button of the main toolbar.

5.14.2. Peak filter

The system provides possibility to filter peaks. This feature is used to search for TIN nodes, which Z value differs considerably from adjacent nodes, so the system defines nodes with sharp local peaks. Peaks filtering allows to remove part of nodes, that form the most 'spiky' pyramids.



TIN filtering is performed considering breaklines.

Perform the following actions to filter peaks:

1. **Create** or **load** TIN.
2. Select **Terrain > TIN > Peak filter**. The **Peak filtering** window opens.

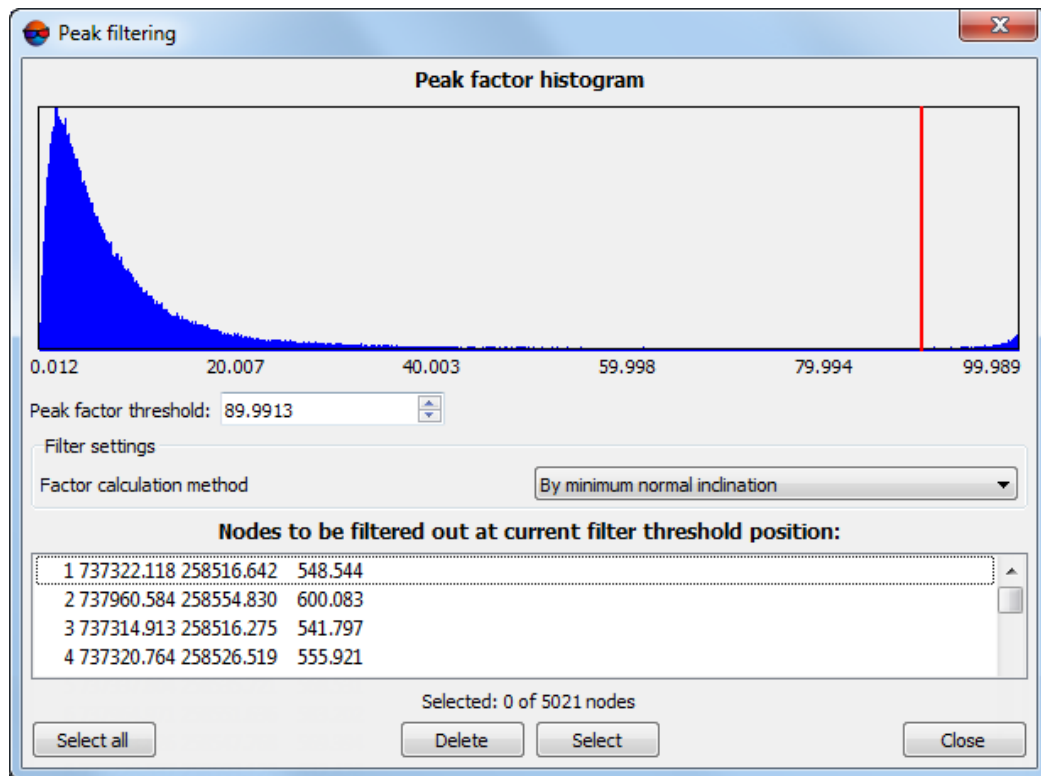


Fig. 63. Peaks filtering

The histogram shows the relative distribution of TIN nodes with different coefficients of peaks. A node with a value of peak factor, that equals 0, is assumed error-free. A node with maximal probability of error (in accordance with selected calculation method) have a peak factor that equals 100.

3. Specify **Peak factor threshold** or move using mouse button the red line on the diagram to filter nodes, that fall into areas with values exceeding specified (behind the red line).
4. In the **Filter settings** section select **Factor calculation method** in the list:
 - **By minimum normal inclination** – peak factor is proportional to the minimum factor by triangles surrounding the vertex, to the value of angle between the normal to the plane of the triangle and Z axis;

- **By maximum normal inclination** – peak factor is proportional to the maximum factor by triangles surrounding the vertex, to the value of angle between the normal to the plane of the triangle and Z axis;
 - **By mean normal inclination** – peak factor is proportional to the mean factor by triangles surrounding the vertex, to the value of angle between the normal to the plane of the triangle and Z axis;
 - **By sum normal projection** – peak factor is inversely proportional to module projection of the vector sum of the normals to triangles surrounding the vertex on Z axis;
 - **By sum normal abs.value** – peak factor is inversely proportional to module projection of the vector sum of the normals to triangles surrounding the vertex.
5. Click the **Refresh** button. The bottom part of the window shows a list of nodes, that correspond to peaks with specified parameters.
 6. Select a node in the list by mouse click. After that marker is placed to the selected node and is displayed in 2D-window.



If the basic vector layer is active and editable, the given vertex is to be selected both in the **Peak filtering** window and in the 2D-window.





To select nodes group in the **Peak filtering** window it is possible to use **Ctrl** or **Shift** keys. The **Select all** button allows to select all nodes found.



To select a group of vertices in a 2D-window, use the **Select** button.

7. Click the **Delete** button to delete points from base vector layer selected in the list. After that the TIN is rebuilt automatically.



Deleting of TIN nodes means removing of vector points from base vector layers. To cancel thinning results the **Undo log** is used, as well as the  and  button of the main toolbar.

5.14.3. TIN filtering by Z-range

The system provides possibility to edit TIN using Z-filter. During filtering by Z-range the system performs deleting of vector points, and polylines/polygons vertices (that correspond to TIN nodes), which Z-coordinate is out of the range.



TIN filtering is performed considering breaklines.

To perform filtering by Z-range of vector objects of base vector layers, used for TIN creation, perform the following actions:

1. **Create** or **load** TIN.
2. Select **Terrain › TIN › Filter › Filter by Z-range**. The **Z-range filter** window opens.

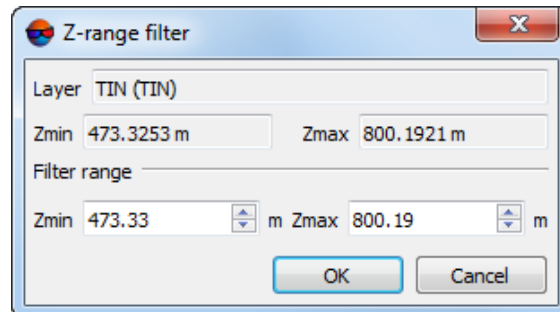




Fig. 64. Parameters of TIN filtering

The **Layer** field displays a name of loaded TIN layer, the **Zmin** and **Zmax** fields show terrain elevation difference in meters, calculated by all TIN nodes.

3. The **Filter range** section by default shows values of calculated terrain elevation difference. Specify maximal and minimal Z values in meters to consider for points filtering.
4. Click OK. Filtration removes all points and vertices of polylines/polygons, which Z-coordinate falls outside specified range.



Deleting of TIN nodes means removing of vector points from base layers. To cancel thinning results the **Undo log** is used, as well as the  and  button of the main toolbar.

5.14.4. Smooth TIN interpolation

The system provides possibility to interpolate TIN network for DEM smoothing for creating or improving contour lines, if contour lines layer was one of the base layers for TIN creation.



TIN filtering is performed considering breaklines.

Perform the following actions to perform TIN interpolation:

1. **Create** or **load** TIN.



To interpolate TIN within a grid it is necessary to create or load the grid **prior to** interpolation operation.

2. Select **Terrain › TIN › Interpolate**. The **Smooth TIN interpolation** window opens.

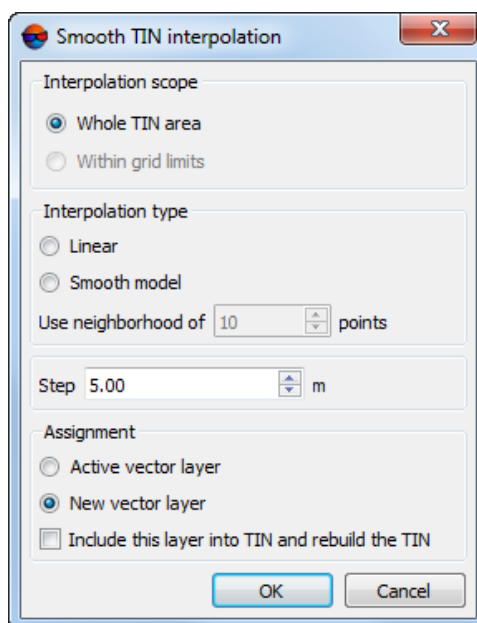


Fig. 65. Parameters of TIN interpolation

3. Specify the **Interpolation scope**:

- **Whole TIN area**;
- **Within grid limits** – the system uses parameters of regular grid with specified step.

4. Specify the **Interpolation type**:

- **Linear** – TIN condensation using points adding on triangles surface;
- **Smooth model** – using second-order surface for interpolation by specified number of points in the neighbourhood of the triangle, where a grid node is located.

For smooth model specify a number of points used to perform interpolation in the **Use neighbourhood of..points** field.



It is recommended to specify a value not exceeding 100 points in the **Use neighbourhood of..points** field.

5. Set the **Step** of interpolation in meters.

6. Select a destination of interpolation results save:

- **Active vector layer** – the active vector layer will be rewritten;

- **New vector layer** – interpolation results will be copied to a new vector layer.

The **Include this layer into TIN and rebuild the TIN** checkbox is set on by default, that allows to update TIN network automatically according to filtering results.

7. Click OK. After that the system performs interpolation of base vector layer and displays results depending on selected parameters.

5.15. Converting TIN into a vector layer

The system provides possibility to convert TIN into a vector layer.

To do this, perform the following:

1. Create or download a TIN layer;
2. Select **Terrain > TIN > Convert to vector layer**. The **Convert TIN** window opens;

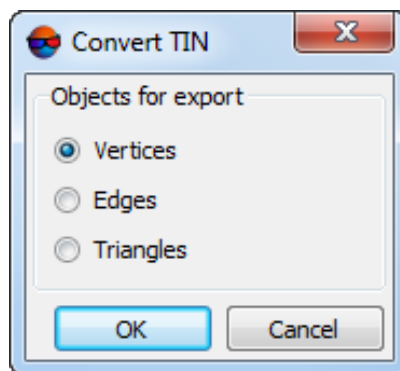


Fig. 66. The Convert TIN window

In the **Convert TIN** window specify the parameters of TIN conversion into a vector layer:

- **Vertices** – to save vertices of irregular triangles as points in the new vector layer;
- **Edges** – to save edges of irregular triangles as polylines in the new vector layer;
- **Triangles** – to save irregular triangles as polygons in the new vector layer;

3. Click OK.

5.16. Creation of textured TIN 3D surface

The system allows to create textured 3D-TIN surfaces. A DEM or LAS point cloud can be used as initial data for creating a 3D surface. Texturing of the created surface is carried out using images of the loaded project (optional, taking into account the color

of LAS input points). The initial DEM or point cloud is to describe the areas of the earth's surface covered by the project images.



The output files of textured 3D-TIN surfaces have *.json extension. For further work with 3D-TIN, the *PHOTOMOD 3D-Mod* program is used (see the “[Three-dimensional modeling](#)” User Manual). 3D-TIN tiles (with *.b3dm extension) are saved in the separate directory (with the same name) in the folder chosen for output 3D-TIN saving in the active profile's resources.

To create the textured 3D-TIN surface, perform the following:

1. [optional] If only a share of the project's images are intended to be used for 3D surface texturing, first select the necessary images in the **Block scheme** window (or in the **Block editor** window);
2. Choose **Terrain > TIN > Build 3D-TIN...**. The **3D-TIN creation** window opens;

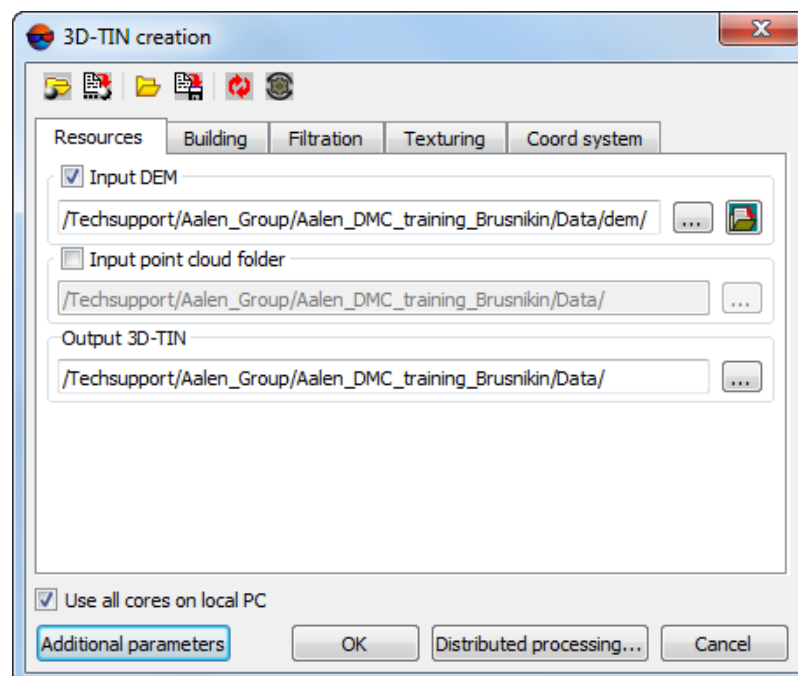






Fig. 67. The 3D-TIN creation window

The **3D-TIN creation** window contains the following elements:

- A toolbar in the upper part of the window, which is intended to save/load parameters settings on all tabs of the window 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.
- The **Resources**, **Building**, **Filtration**, **Texturing** and **Coord System** tabs;
- The **Use all cores on local PC** checkbox which allows to use all the cores of workstation CPU for calculation operation;




It is necessary to clear the checkbox to use only one core.

- The **Additional parameters** button which allows to set additional parameters of 3D-TIN creation in **Building** and **Texturing** tabs.

3. Select the input **Resources** in an appropriate tab:

- [optional] **Input DEM** – click the  button to select input DEM in active profile resources;



Click the  button to open the **Layer selection** window and select the source DEM from the list of DEMs uploaded to the project.

- [optional] **Input point cloud folder** – click the  button to select the folder with input point clouds (LAS) in active profile resources.



The input point clouds (LAS) should be [created](#) in *PHOTOMOD* system.

The third-party lidar data could be used as input point cloud only after the point cloud colorization by the current project images (see the “Point cloud colorization by project images” chapter of the “[LIDAR Data processing](#)” User Manual).

Input path to save file to the **Output 3D-TIN** field or click the  button, to select name and path in active profile resources to save 3D-TIN.

- ### 4. [optional] Set the following main and additional 3D-TIN **building** parameters in an appropriate tab:

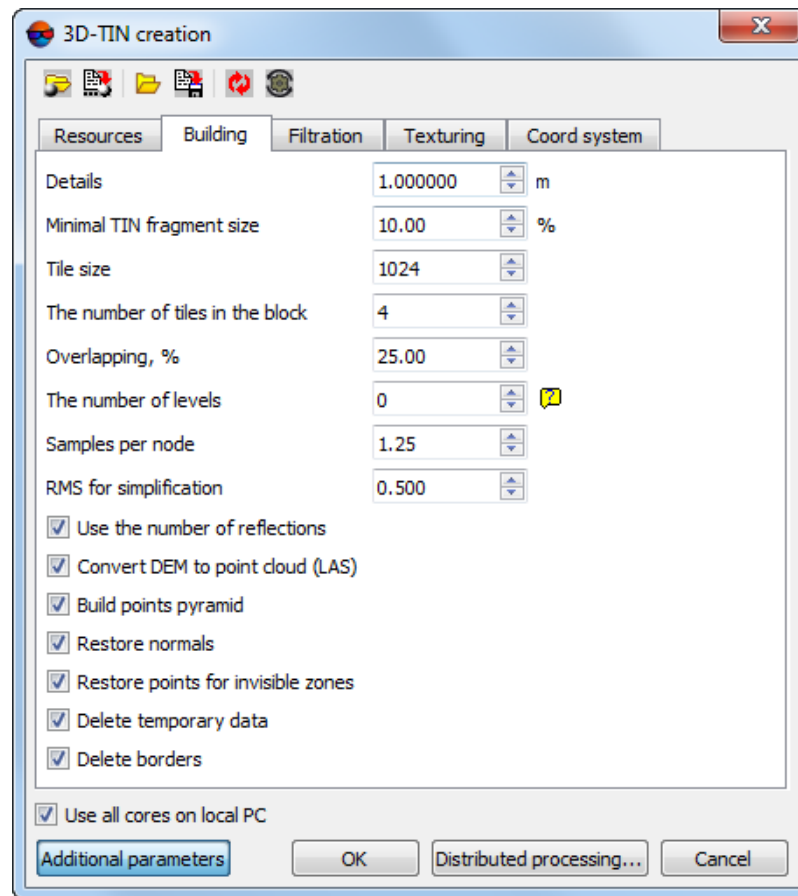


Fig. 68. The Building tab

Set the main 3D-TIN **building** parameters:

- The **Detailing** parameter (set in the units of the current project) allows to set the detail degree of created 3D-TIN:
 - If the **Input DEM** is used, the recommended value of this parameter corresponds to the input DEM cell size (GSD). To build a 3D-TIN with a higher level of detail is possible but not advised (an appropriate warning is issued);
 - If the **Input point cloud folder** is used, it is recommended to input a value that exceeds the **input point cloud** resolution 2-3-fold (i.e. the resolution is to be 2-3-fold worse than the input point cloud resolution).



The quality of 3D-TIN building directly depends on the quality of the **Input DEM** (point cloud), resolution of source project imagery used for texture creation, as well as on the suitable interrelation between 3D-TIN detailing and the parameters that determine texture resolution (see the **Texturing** tab description below).



For nadir imaging with medium ground sample distance (GSD) more than 0.1 m (that doesn't depict small parts of buildings), it is recommended to use an **Input DEM**.

For the projects containing higher quality original imagery (GSD < 0.1 m), it is recommended to use a high resolution input point cloud.



If maximum detailing is needed, use an **Input DEM** whose cell size corresponds to one image pixel.

In this case:

- to create an extra detailed 3D-TIN, the **Detailing** option value is to outnumber the **input DEM** cell size 2-3-fold;
- to create a detailed 3D-TIN, the **Detailing** option value is to outnumber the **input DEM** cell size 4-6-fold;
- to create a survey 3D-TIN, the **Detailing** option value is to outnumber the **input DEM** cell size ~10-fold.

- [optional] to **Delete borders** of the input point cloud set an appropriate checkbox;



This function is available only if the source for 3D-TIN building is the **Input point cloud folder**. To **convert DEM to point cloud (LAS)**, set the appropriate checkbox (**Building** tab, **Additional parameters**).



Leave the **Delete borders** checkbox set. Otherwise, the quality of 3D-TIN boundary regions' construction may decrease.

[optional] to set the **Additional parameters** of 3D-TIN building click the appropriate button:

- In the **Minimum TIN fragment size** input field, set minimum size of isolated 3D-TIN areas, that are not to be deleted;



Isolated 3D-TIN areas – are triangle groups that are not directly related with the main 3D-TIN surface and can arise, including due to errors. Isolated areas are subjected to filtering at final stages of 3D-TIN creation. The filtering criterion is the size of the isolated area (number of triangles).

The size of an isolated area (number of triangles in the isolated area) is compared with the size (number of triangles) of the particular tile containing the isolated area being tested. The **Minimum TIN fragment size**, which determines the stringency of filtering, is set as a percentage of the number of triangles in the tile (see below).

- In the **Tile size** field, customize sizes of 3D-TIN tiles by setting the approximate number of triangles in individual tile;



The product of the values of the **Tile size** and the **Detailing** parameters approximately corresponds to the tile size in the units of the project.

- In **The number of tiles in the block** field, set the number of tiles to be processed in the framework of a single task;



Reducing **The number of tiles in the block** leads to the increased number of distributed processing tasks and increased 3D-TIN building time. It is advisable to reduce **The number of tiles in the block** in case of a lack of system resources.



Increasing **The number of tiles in the block** leads to the decreased number of distributed processing tasks what may improve the system performance in case of available system resources.

- In the **Overlapping** field, set the 3D-TIN fragment overlap, in %;



It is not recommended to set null overlaps.

- In **The number of levels** field, set the number of levels of 3D-TIN pyramid;



The system provides the capability to create multi-level 3D-TINs that have various detailing upon review depending on the degree of approximation (when **the number of levels** of the pyramid is more than 1, otherwise if looked up in *PHOTOMOD 3D-Mod* 3D-TIN would be open with the only maximum detailing).



The null value allows to calculate **The number of levels** of a 3D-TIN pyramid automatically.

- The **Use the number of reflections** checkbox allows one to take into account the number of **reflections** per point when building a 3D-TIN (i.e. points with a large number of reflections will have more weight when processing the input point cloud).



This function is available only if the source for 3D-TIN building is the **Input point cloud folder**. To **convert DEM to points**, set the appropriate checkbox (**Building tab, Additional parameters**).



In case of the **cloud of points, created during the DEM creation using the SGM method**, the number of reflected pulses in the given point is the number of stereopairs on the basis of which a particular point was calculated: 1, 2, 3, 4, 5, 6, 7, or 8.

Limitation of the “number of reflected pulses” by 1 to 8 results from the limitations of the LAS format itself, accordingly, in the latter case, the number of stereopairs can be both “8” and “8 and more”.

- Configure the **Samples per node** parameter;



This function is available only if the source for 3D-TIN building is the **Input point cloud folder**. To **convert DEM to points**, set the appropriate checkbox (**Building tab, Additional parameters**).



The display of separate objects on the constructed 3D-TIN surface depends not only on the quality of the source data, but also on the user-defined level of detail of the output data (see above).

For example, an object with plane dimensions smaller than or comparable to the specified output detail may not be displayed on the constructed 3D-TIN surface.

By increasing the **Samples per node** parameter, the user can select the appropriate settings for displaying objects with small linear dimensions, taking into account the detail of the output data.

The approximate value of this parameter is the ratio between the input data GSD and the minimum linear dimensions of the objects that must be displayed when building a 3D-TIN.



The increase of this parameter causes the increase of time required for 3D-TIN building.

- Enter the **RMS for simplification** value;



This function is available only if the source for 3D-TIN building is the **Input point cloud folder**. To **convert DEM to points**, set the appropriate checkbox (**Building** tab, **Additional parameters**).



When creating a 3D-TIN surface, the initial detailed surface, built in accordance with the characteristics of the source data, is smoothed and simplified according to the user-specified building parameters (see above).

Thus, for example, the use of an excessive number of triangles to describe a part of an object that is a plane (or close in shape to a plane) is somewhat redundant.

By changing the limiting value of **RMS for simplification** of the surface, the user may adjust the stringency of algorithms leading to a decrease in the number of triangles that describe the 3D-TIN surface.



The default (0.25) value is recommended.

- To **convert DEM to points**, do not clear the appropriate default checkbox before 3D-TIN building.



Some 3D-TIN building parameters (see above) are available only in case when the source for 3D-TIN building is a point cloud.

- To **build points pyramid** before building a 3D-TIN, do not clear the appropriate default checkbox.



This function is available only if the source for 3D-TIN building is the **Input point cloud folder**. To **convert DEM to points**, set the appropriate checkbox (**Building** tab, **Additional parameters**).



It is recommended to **build points pyramid** before building a 3D-TIN. Otherwise, the time spent on building a 3D-TIN may increase significantly.

- To **restore points for invisible zones**, set the appropriate checkbox.



The system allows for calculating and building sections of a 3D surface that describe elements of objects on the ground that are not displayed in the project images, for example, such as steep walls of buildings that appear on all project images in the so-called “occluded zone”. It is recommended to **restore points for invisible zones** if images with small off-nadir angles were used when project-creating, which generally display the same-type housing development with simple architectural concepts that do not contain complex geometric shapes.

- To **restore normals**, set the appropriate checkbox.



The directions of normals at points in the input cloud are taken into account when approximating the 3D surface. The system allows one to clarify the directions of the specified normals (by analyzing normal directions at neighboring points). This has a positive effect on the output 3D surface construction quality. In most cases, it is not recommended to clear this checkbox.

- To **delete temporary data** when building a 3D-TIN from a point cloud, set the appropriate checkbox.



Temporary data that arise when building a 3D-TIN from a point cloud (i.e. if the initial resource for building is the **Input point cloud folder**, or the **convert DEM to points** checkbox is set) can be quite large.

However, stored temporary data can significantly improve system performance if the operation is repeated with the same initial data (e.g. if the first-time result was not considered satisfactory and a decision was made to re-build 3D-TIN with adjusted building parameters).

If other input data are used (or if the **convert DEM to points** checkbox was subsequently cleared), the stored temporary data will not play any role in the 3D-TIN construction.

5. In the appropriate tab, set the parameters for 3D-TIN **filtration**.



In this case, by filtering we mean 3D-TIN smoothing due to clarifying positions of vertices without their removal.

The system allows for filtering both before and after the simplification of the created model according to preset **detailing** parameters (see the **Building** tab). In both cases, the **bilateral**, **smooth** and **clarify** filters can be applied during the filtering any times in the order selected by a user. Every above filter application, and their order are customized individually by a user, in the appropriate lists in left and/or right part of the window.

In the list from the left part of the window, user can customize the parameters and sequence of filters applied for pre-filtering of a non-simplified 3D-TIN model. The list in the right part of the window allows to customize repeated filtering for a 3D-TIN model simplified according to preset detailing parameters. The user has the opportunity not to perform preliminary and/or repeated filtering of 3D-TIN at all by clearing the appropriate checkboxes or removing filters from the corresponding lists in the left or right part of the window. However, to ensure suitable 3D-TIN quality, it is highly recommended to perform filtering.

Pre-filtering is the most preferable towards maximum 3D-TIN model accuracy, but it is extremely resource-demanding operation if the **Bilateral filter** is applied. The **Smooth filter** is less system resource-demanding, however its application is associated with significant time spending. Bilateral and/or smoothing filters applied to pre-simplified modes (without pre-filtering) allows to significantly increase system performance, thus avoiding high time spending or involving significant computational resources.



The **Bilateral filter**, in most cases, ensures fast creation and high quality of an output 3D-TIN. However, as it was noted above, it is very resource demanding if used for pre-filtering. In this case, when creating a very detailed 3D-TIN surface, it is highly recommended to decrease **Tile size** down to 32 (see above), and **Max MT tasks** for a workstation with a multicore processor and 16 Gb RAM - down to 1 (see distributed processing parameter settings below).

The **smooth filter**, under similar circumstances, is less workstation resource-demanding than the bilateral one, has more flexible settings and provides a comparable quality of data processing, however, when it is used for pre-filtering, the time spent on building a 3D-TIN significantly increases. If to skip pre-filtering, and apply filters only to a simplified model, the differences described above are not so significant.

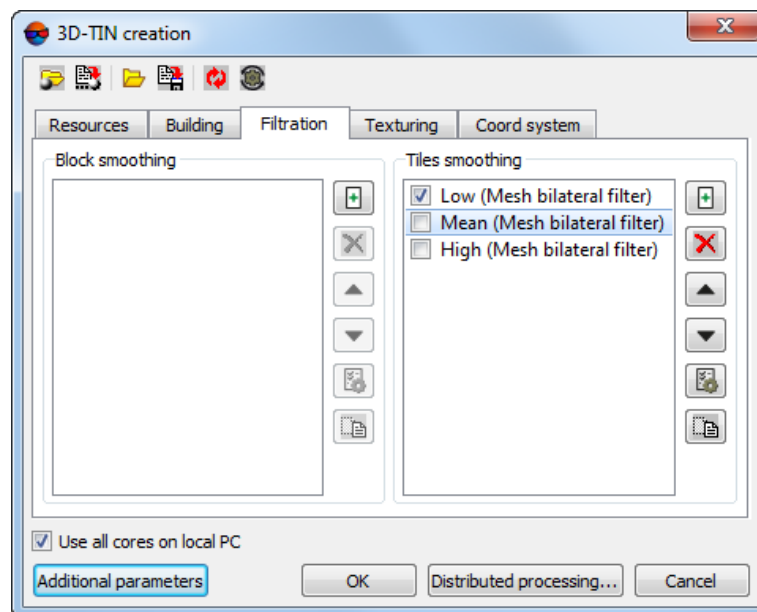



Fig. 69. The Filtration tab

The toolbar of the **Filtration** tab contains the following buttons:

-  – allows to add the **bilateral**, **smooth** or **clarify** filter into the set of filters providing pre-filtering or simplified model filtering;

-  – removes the selected filter from the set;








Select a filter by clicking the left mouse click.




To exclude a filter from processing not removing it, clear the appropriate checkbox.



The system provides for using pre-configured filters suggested by default. To restore a deleted filter from this list, click the  button.

-  – allows to set the order of filters by moving the selected filter higher in the list;
-  – allows to set the order of filters by moving the selected filter lower in the list;
-  – allows to configure the selected filter;
-  – allows to rename previously added filter.

To add the **Bilateral filter** into the set of filters providing pre-filtering or simplified model filtering, perform the following:

1. [optional] click the  button in the left or right part of the window. The **Add filter** window opens:

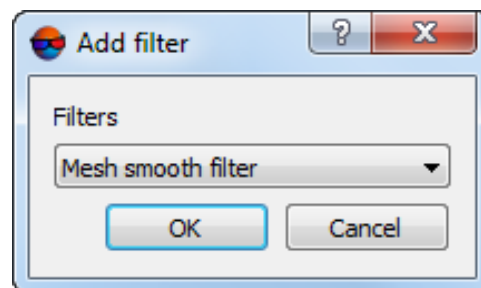


Fig. 70. The Add filter window

2. Select the **Bilateral filter** by clicking the left mouse button and click Ok;
3. Input filter name in appropriate window:

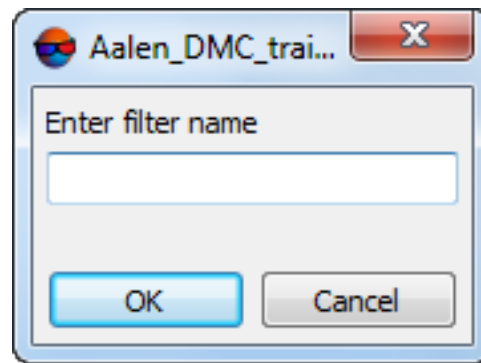


Fig. 71. The input window

4. [optional] Select added **bilateral filter** by clicking the left mouse button and click the  button. The **Bilateral filter** window opens:

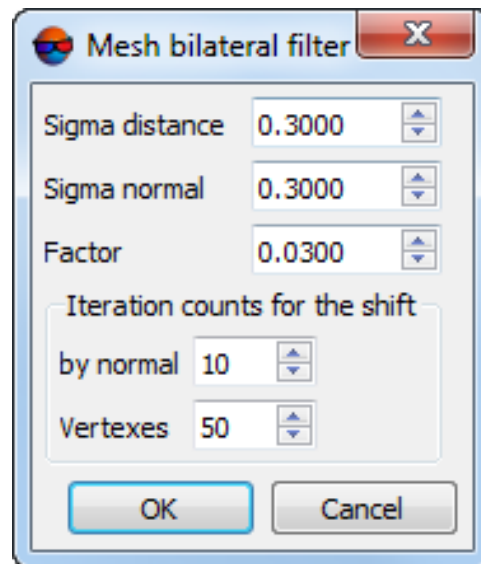


Fig. 72. The Bilateral filter window



The **bilateral filter** is used to clarify the position of 3D-TIN vertices without their removal. The filter is based on the analysis of the relative position of normals to 3D-TIN faces. Corrections to 3D-TIN vertices are accumulated through the sequence of iterations during those there are calculated total weight normals to each face adjacent to the processed vertex. Corrections are applied in several stages (vertex shift). Ideally, in case of quite long-term filter operation, the vertex shift in the filtered 3D-TIN at each next iteration, beginning from the appropriate moment is to be minimal.

When configuring a bilateral filter, set the following parameters:

- A range for search of neighbor faces (in the vicinity of the analyzed face) is a production of the **Detailing** (see above) and **Sigma distance** parameters;



Sigma distance directly affects the number of neighboring faces used when analyzing a single face under processing. Significant increase of this parameter against the default value may cause essential loss of performance, especially when creating detailed 3D-TIN.

- The **Sigma normal** that determines the degree of impact of neighboring face normals (within a search range) on the calculation of a total weight normal to the analyzed face.



The **Sigma normal** is set between the zero (null impact) and the unity (max impact). When normal sigma values are close to the zero, the **bilateral filter** will not correct the existing roughness of a model, and high values will cause geometric data distortion (for example, rounded building corners).

- An **iterations count for the shift by normals** during the calculation of corrections to vertices;
- The **factor** of influence of accumulated corrections on the positions of 3D-TIN vertices when they are shifted;





When processing data that display terrain with significant development, it is not recommended to exceed the default **factor** of influence.

- An **iterations count for the shift by vertex**.

Click Ok.

5. [optional] Select added **bilateral filter** by clicking the left mouse button and change its position using the ▲ and ▼ buttons, if appropriate.

To add the **Smooth filter** into the set of filters providing pre-filtering or simplified model filtering, perform the following:

1. [optional] click the  button in the left or right part of the window. The **Add filter** window opens:
2. Select the **Smooth filter** by clicking the left mouse button and click OK;
3. Input filter name in appropriate window:
4. [optional] Select added **smooth filter** by clicking the left mouse button and click the  button. The **Smooth filter** window opens:

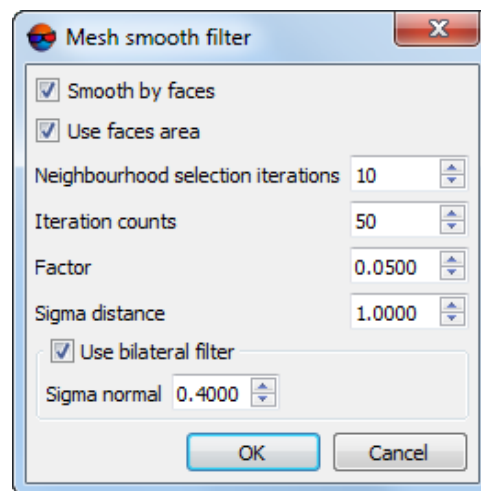


Fig. 73. The Smooth filter window



The **smooth filter** is used to clarify the positions of 3D-TIN vertices without their removal. Corrections to each of the vertices are calculated based on the analysis of mutual position of the neighboring 3D-TIN vertices (or faces).

Set the following parameters of the **smooth filter**:

- [optional] In order to calculate the corrections for the position of vertices based on the results of the analysis of the relative position of the 3D-TIN faces closest to them, set the **Smooth by faces** checkbox, otherwise, the corrections are calculated based on the results of the analysis of the relative position of the nearest vertices.
 - To consider the area of a face (as a weight function) when calculating these corrections, set the **Use faces area** checkbox. Otherwise, the area of a face is not taken into account (each face has the same weight at correction calculation).
- Configure the parameters for search of the closest vertices (or faces) in the vicinity of a vertex under processing:
 - The number of **neighbourhood selection iterations** is max neighbor search depth, i.e. search of faces (or vertices) directly osculant to the vertex under processing (at the first step) and further, search of faces or vertices osculant to the “neighbors”.



The number of **neighbourhood selection iterations** directly impacts the number of adjoining faces (vertices) used for the processing of the current vertex. Significant increase of this parameter against the default value may cause essential loss of performance, especially when creating detailed 3D-TIN.

- A range for search of adjacent faces (vertices) in the vicinity of the analyzed face is a production of the **Detailing** (see above) and **Sigma distance**;



Sigma distance additionally limits the number of **neighbourhood selection iterations**.

- The **factor** of influence of correction to the 3D-TIN vertex position;





When processing data that display terrain with significant development, it is not recommended to exceed the default **factor** of influence.

- Set the iterations counts for filter operation;
- [optional] to use also the **bilateral filter**, set the appropriate checkbox and set the **Sigma normal** parameter (see above).

Click OK.

5. [optional] Select added smooth filter by clicking the left mouse button and change its position using the ▲ and ▼ buttons, if appropriate.

To add the **Clarify filter** into the set of filters providing pre-filtering or simplified model filtering, perform the following:

1. [optional] click the  button in the left or right part of the window. The **Add filter** window opens:
2. Select the **Clarify filter** by clicking the left mouse button and click Ok;
3. Input filter name in appropriate window:
4. [optional] Select added **clarify filter** by clicking the left mouse button and click the  button. The **Clarify filter** window opens:

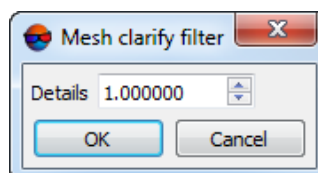


Fig. 74. The Clarify filter window

Specify the detalization settings.

Click Ok.

5. [optional] Select added **clarify filter** by clicking the left mouse button and change its position using the ▲ and ▼ buttons, if appropriate.

6. [optional] In the **Texturing** tab, set the **Build textures** checkbox to create a textured 3D-TIN surface:

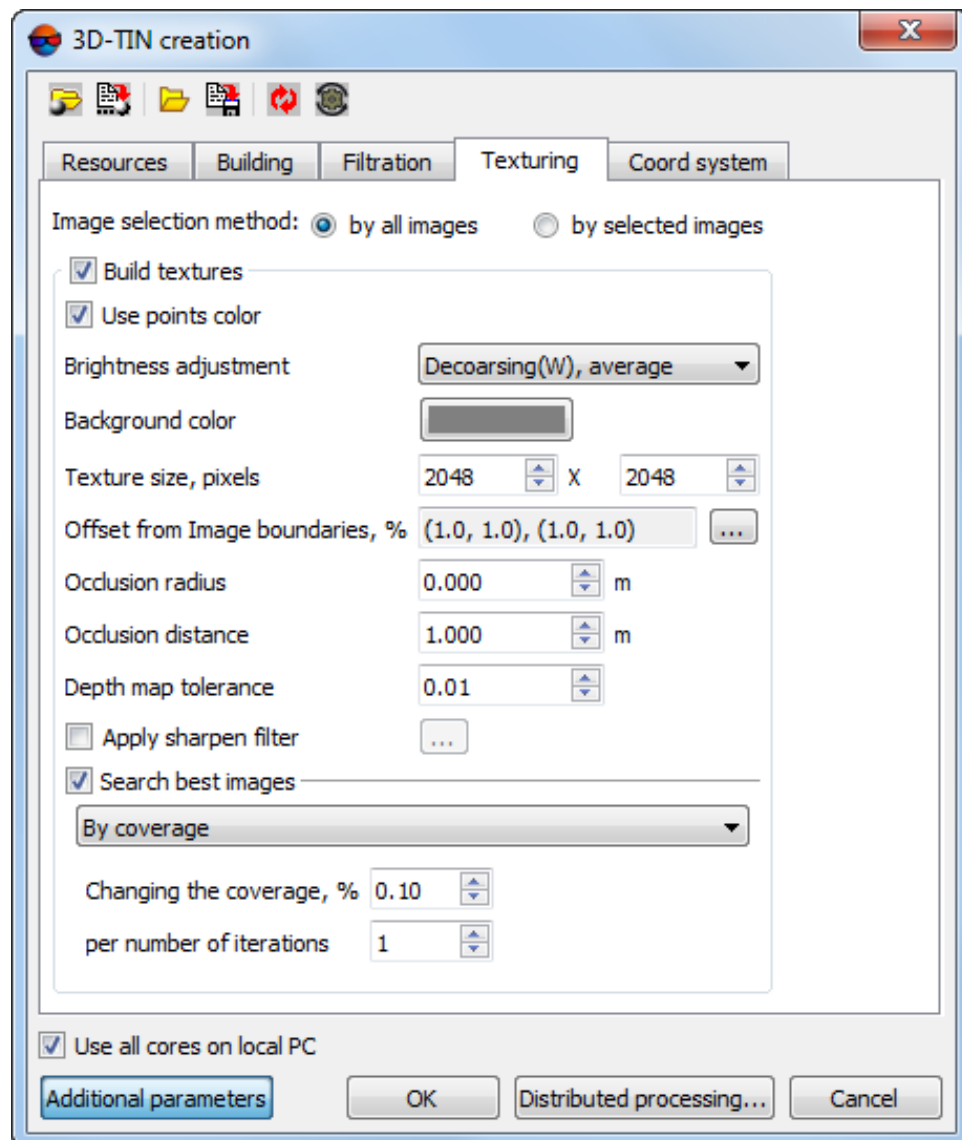


Fig. 75. The Texturing tab

Set the main parameters of the textures creation:

- Specify the **image selection method**:
 - **by all images**;
 - **by selected images**.



The **image selection method** can have a significant impact on the **brightness adjustment** between the textures of 3D surface fragments (see above), especially if images with noticeably different spectral and brightness characteristics (for example,

taken with a significant time interval, e.g., in different seasons, under different lighting conditions, etc.) are used for project processing.

- [optional] To **use points color** of the input cloud points when creating textures (if such data is available in the input LAS files), set the appropriate checkbox;



This option is available only if the **input point cloud folder** is used as input data (see the **Resources** tab).



Particularly, the color of neighboring points can be used when calculating the final colors of pixels in a 3D surface region that describes fragments of objects that are not displayed in the project images (located in occluded zones in all images). To use a uniform user-defined **background color** (see below) when texturing similar 3D surface elements, clear this checkbox.

- Choose the **brightness adjustment** method for equalizing brightness between textures of 3D surface fragments:
 - **Best;**
 - **Average;**
 - **Decoarsing, best;**
 - **Decoarsing, average;**
 - **Decoarsing, nearest;**
 - **Decoarsing (W), best;**
 - **Decoarsing (W), average;**
 - **Decoarsing (W), nearest.**



All images covering a 3D surface fragment under processing are involved in the color calculation. **Brightness adjustment** allows a user to create a visually unified and continuous 3D surface by smoothing out differences between the brightness and contrast characteristics of its individual fragments.

The proposed techniques for calculating the color of the final pixel of a 3D surface generally come down to either using the average brightness and contrast characteristics of the relevant pixels of all images involved or to using the color of a pixel of a specific image that best satisfies certain conditions, i.e., maximum proximity to the projection center and the smallest off-nadir angle at a given point on the 3D surface.

Averaging the characteristics of all involved images usually leads to the creation of visually continuous 3D model textures devoid of sharp transitions between areas with different color, brightness, and contrast characteristics. A disadvantage of such a technique is the possible blurring that may occur in certain circumstances in some areas of 3D surface textures.

The use of brightness characteristics of individual images recognized as the most suitable when calculating a specific pixel, in turn, leads to the greatest reliability when creating textures for individual fragments, negatively affecting, in certain cases, the quality of brightness adjustment across the entire 3D surface as a whole.

Automatic filtering allows users to pre-exclude from processing those images in which the brightness and contrast characteristics of the area of interest differ significantly from the majority of images covering the given area (for example, if a particular image was created under changed lighting conditions).

The user also has the option to preliminarily select images intended for creating textures, manually rejecting unsuitable images (see below).

- Select the **background color** used when texturing fragments of the 3D surface that are not displayed in any of the project images used (being in so-called “occluded zones”);
- Set the **Texture size**, in pixels;



It is recommended to set the **Texture size** exceeding the **Tile size** (see above) at least twice (or more). Increasing of this parameter results in increased texture resolution and increased time of 3D-TIN building:

- If the **Tile size** is 64, the recommended **Texture size** is 256.
- If the **Tile size** is 32, the recommended **Texture size** is 128.




If the *Remote Desktop Protocol (RDP)* is used for the access to the workstation with *PHOTOMOD* installed, then to ensure correct visual assessment of texturing results, it is strongly recommended to set the **Texture size** not exceeding 1024 pixels and divisible by powers of 2 (2, 4, 8, 16, 32, 64, 128, 256, 512, 1024).

Otherwise, due to the peculiarities of the *Remote Desktop Protocol*, correct display of output 3D-TIN textures on the screen of a computer remotely connected to a workstation with *PHOTOMOD* installed is not ensured.



RDP is the *Remote Desktop Protocol* supported by *Microsoft* used to work remotely with another computer. This technology allows for controlling a remote computer at a distance, from the user's workstation, using an Internet connection.

- In order not to use boundary regions of imagery while texturing, 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 texturing;



It may be necessary so that only useful part of an image was textured (e.g. while processing projects with analog imagery as source data).

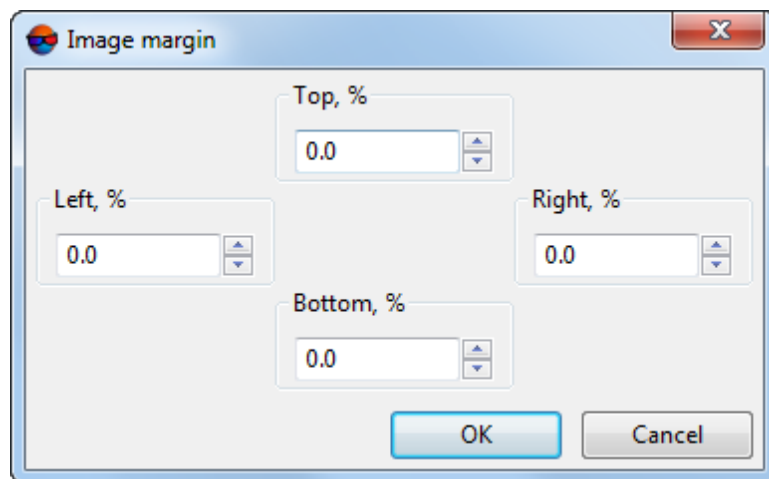


Fig. 76. The Image margin window



Fig. 77. Offset from image boundaries, %

- Set the **Occlusion radius** in pixels when texturing overlapping objects;



Recommended **Occlusion radius** is several pixels. If this parameter is set to zero, invisible areas will not be checked what may result in incorrect texturing of overlapping objects (especially when processing projects with “oblique” survey data as source data).

- Set the **Occlusion distance** – minimum distance to invisible areas in the project units (i.e. minimum distance between overlapping objects). Invisible areas will be verified when this distance is exceeded.



Invisible area check will not be performed if the **Occlusion radius** (see above) is equal to zero.

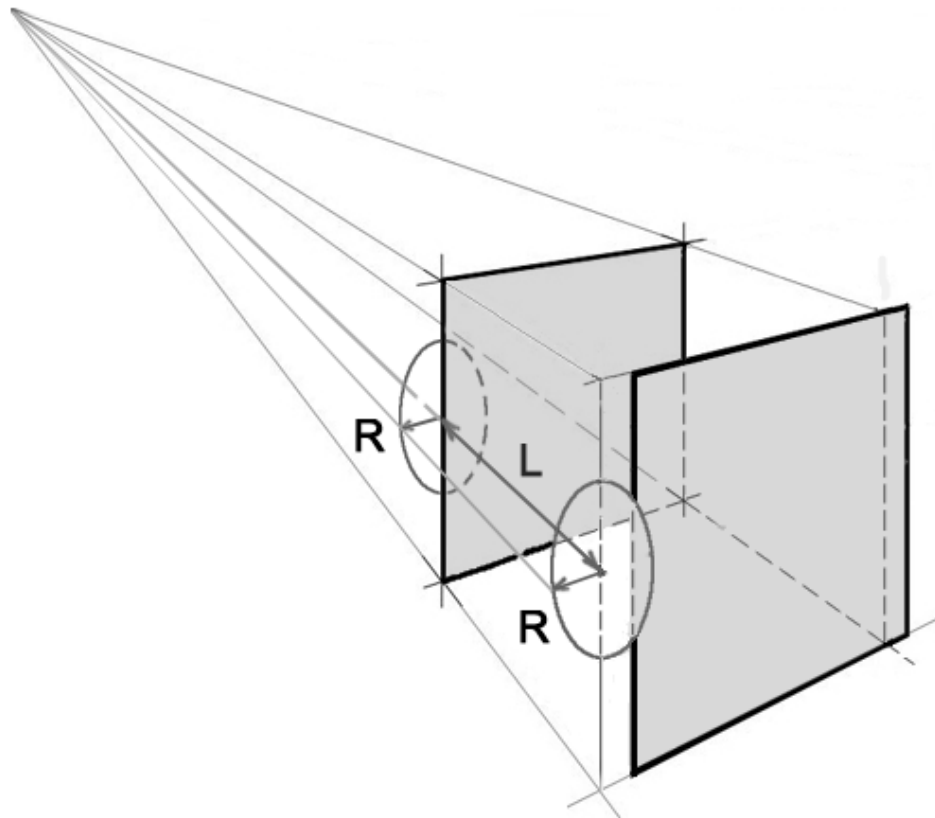



Fig. 78. Invisible areas when texturing overlapping objects (hatched), where R is the **Occlusion radius** and L is the **Occlusion distance**

[optional] to set the **Additional parameters** of textures creation click the appropriate button:

- Set the **Depth map tolerance** in the project units. The **Occlusion distance** is calculated on the base of the depth map;
- [optional] set the **Apply sharpen filter** 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.

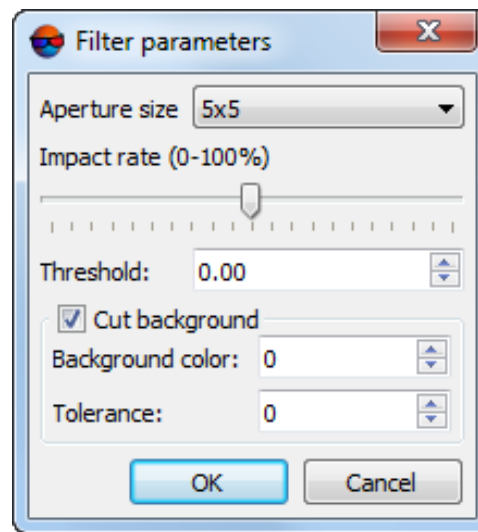


Fig. 79. Filter parameters

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**.



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] By default the **Cut background** checkbox is set on for removing background on image edges. To remove background set the following parameters:

■ **Background color** – value of background color for cutting;



If the 0 value is set in the **Background color** field, background does not consider during correction.

■ **Tolerance** – deviation from background color value for cutting.

- Click OK to close the window.
- [optional] The system allows the user to speed up the process of 3D-TIN texturing by limiting the number of images used. To do this, set the **Search the best images** checkbox and select the criterion for terminating the selection process:
 - **By coverage**;

- Set minimum coverage change (in percent) for a certain number of iterations;
- **By iterations.**
- Set maximum **number of iterations**;



For each 3D-TIN fragment, the selection of the best images is performed. If after the texturing process completion there are areas unfilled with textures, the selection of the best images (from among the unused ones) is repeated, either until the maximum user-defined number of iterations specified is reached, or until the process loses its efficiency, according to the criteria specified by the user (minimum texture coverage change for the specified number of iterations).

7. [optional] In the **Coord system** tab, set the **Convert CS** checkbox to transform 3D-TIN from one coordinate system to another:

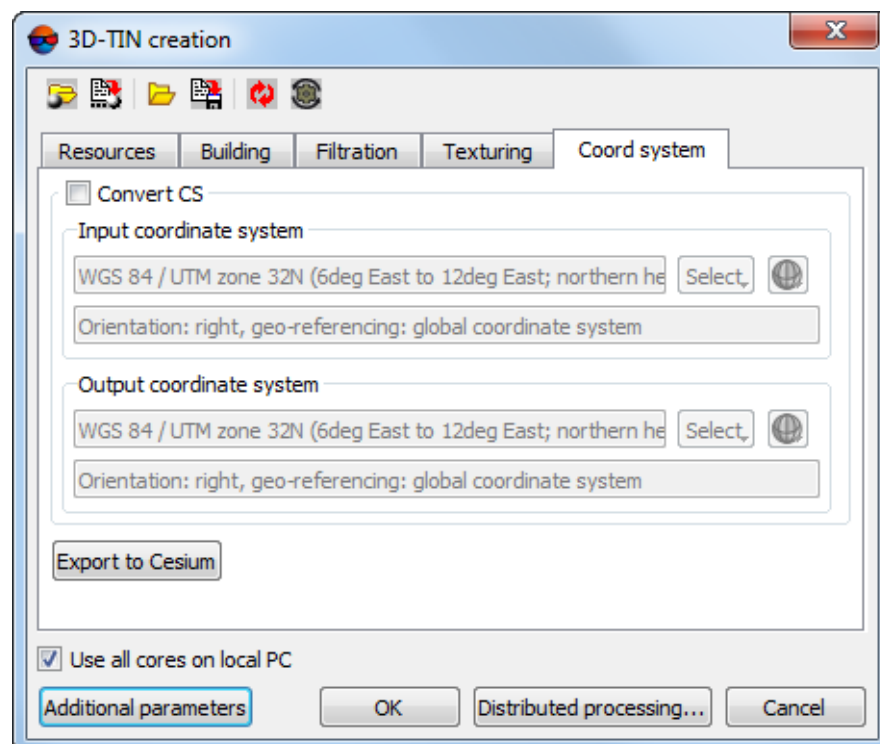


Fig. 80. The Coord system tab

- In the **Input coordinates system** section select actual Coordinate system of input **resources** (if it is different from project coordinate system). Perform the following actions to do this:
 1. Click the **Select...** button to specify initial coordinate system.

Coordinate system is specified using one of the following ways:

- **From DB** – from international or Russian coordinate system database (see "Coordinate systems databases" in the [Project creation](#) User Manual);
- **From file** – allows to select coordinate system from files with *.x-ref-system, extension located out of active profile resources;
- **From resource** – from files with *.x-ref-system extension located in active profile resources, for example, to select coordinate system from another active profile project.
- **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).



The system also allows to select coordinate system from a list of recently used coordinate systems.

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



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

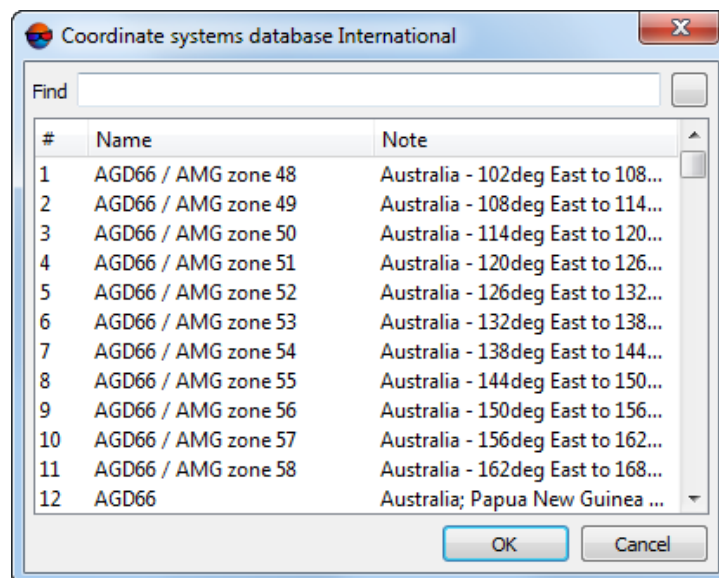



Fig. 81. Window used to select coordinate system from coordinate system database

3. [optional] 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.

- In the **Output coordinate system** define target coordinate system in which it is necessary to convert the 3D-TIN. To do this perform the actions described above or click the **Export model to Cesium** button to select the appropriate coordinate system.



For CESIUM 1.38 version, the coordinate system does not matches the geocentric reference one. It is rotated by -90 degrees around the X axis. CESIUM 1.70 coordinate system matches the classic geocentric reference system with WGS84 ellipsoid and orientation.

8. Click OK to start operation of 3D-TIN creation in usual mode.

In order to create 3D-TIN using distributed processing, 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. 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.

If the **Bilateral filter** is used for pre-filtering, when creating a very detailed 3D-TIN surface, it is highly recommended to set **Max MT tasks** for a workstation with a multicore processor and 16 Gb RAM no higher than 1, and to decrease **Tile size** down to 32 (see the **Building** tab),

3. Click the **Distributed processing** button.



Depending on the size of creation area the operation can take a long time.

5.16.1. Transformation of 3D-TIN coordinate system

The system provides opportunity to transform 3D-TIN from one coordinate system to another.

In order to transform 3D-TIN to another coordinate system perform the following actions:

1. Select **Terrain > TIN > Convert 3D-TIN CS**. The **Convert 3D-TIN to another CS** window opens.

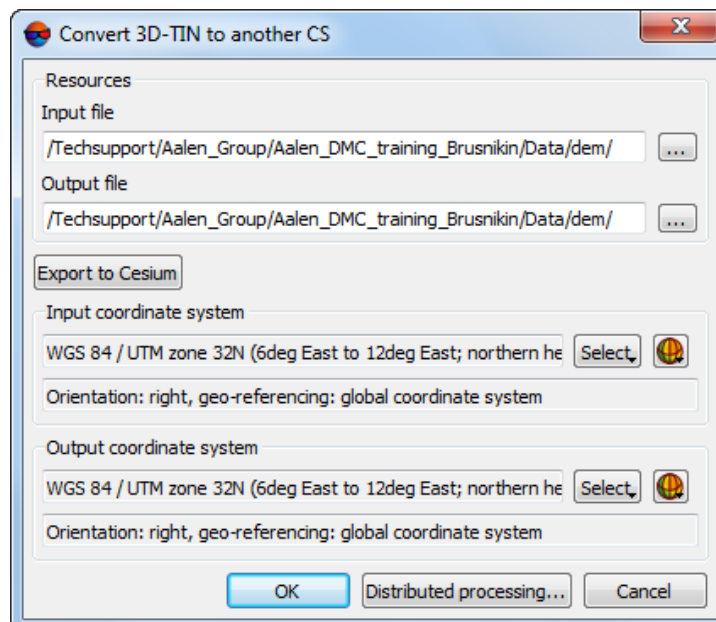




Fig. 82. Transform 3D-TIN to another coordinate system

2. Click the  button in the **Input 3D-TIN** section and specify a source 3D-TIN file in active profile resources.
3. Click the  button in the **Output 3D-TIN** section and specify 3D-TIN name and target folder in active profile resources.
4. In the **Initial coordinates system** section select actual Coordinate system of 3D-TIN (if it is different from project coordinate system). Perform the following actions to do this:
 1. Click the **Select...** button to specify initial coordinate system.

Coordinate system is specified using one of the following ways:

- **From DB** – from international or Russian coordinate system database (see 'Coordinate systems databases' in the [Project creation](#) User Manual);
- **From file** – allows to select coordinate system from files with *.x-ref-system, extension located out of active profile resources;
- **From resource** – from files with *.x-ref-system extension located in active profile resources, for example, to select coordinate system from another active profile project.
- **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).



The system also allows to select coordinate system from a list of recently used coordinate systems.

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



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

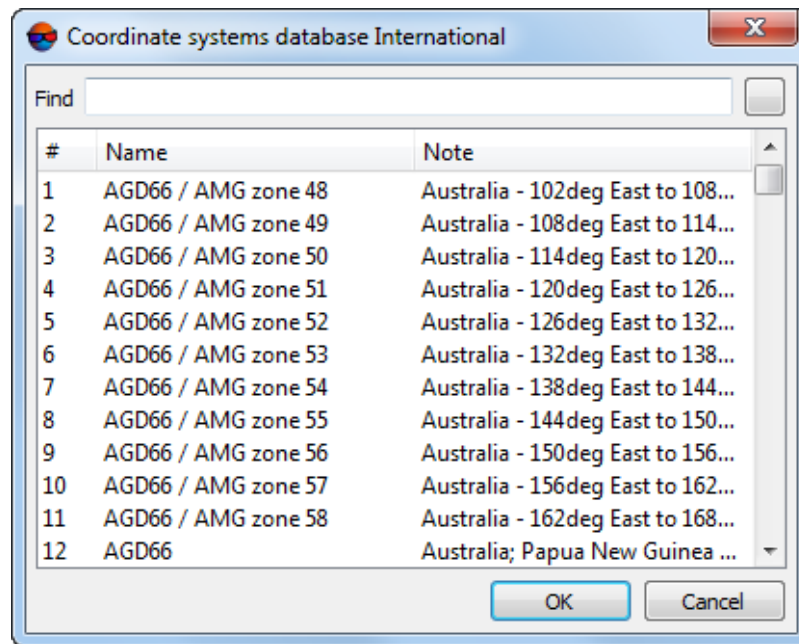



Fig. 83. Window used to select coordinate system from coordinate system database

3. [optional] 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.

5. In the **Output coordinate system** define target coordinate system in which it is necessary to convert the 3D-TIN. To do this perform actions from step 4 or click the **Export to Cesium** button to select the appropriate coordinate system.



For CESIUM 1.38 version, the coordinate system does not matches the geocentric reference one. It is rotated by -90 degrees around the X axis. CESIUM 1.70 coordinate system matches the classic geocentric reference system with WGS84 ellipsoid and orientation.

6. Click OK to convert 3D-TIN coordinate system.

To use distributed computing for changing of 3D-TIN coordinate system, 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.

5.16.2. Batch 3D-TIN creation

The system allows to create textured 3D-TIN surfaces in batch mode.

To create the textured 3D-TIN surface in batch mode, perform the following:

1. Choose **Terrain > TIN > Build 3D-TIN (Batch mode...)**. The **3D-TIN creation** window opens;

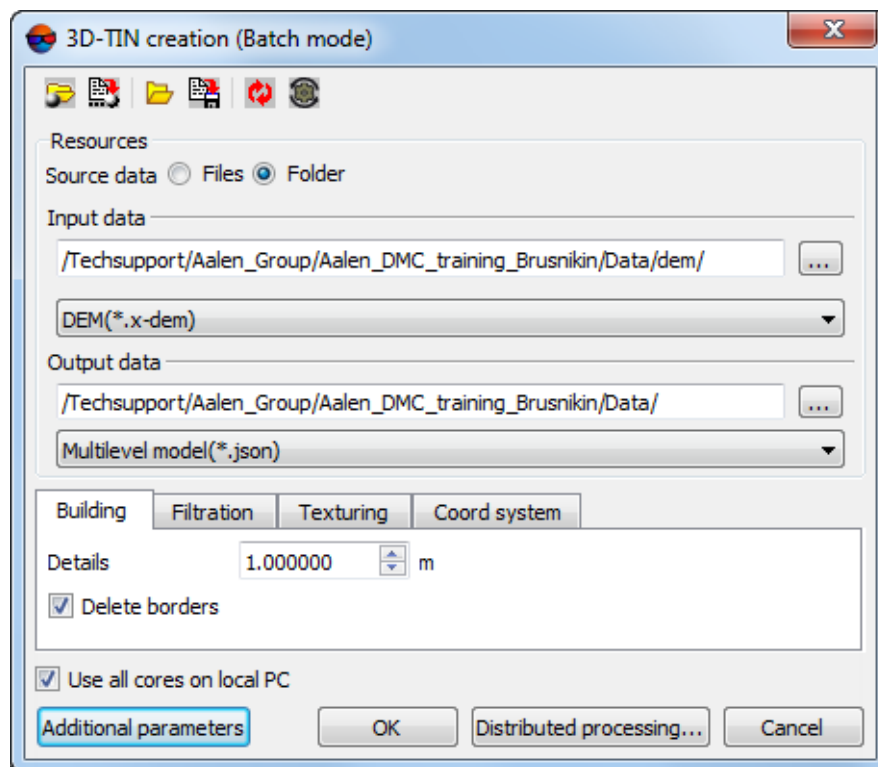








Fig. 84. The 3D-TIN creation window



The **3D-TIN creation** window contains the following elements:


- A toolbar in the upper part of the window, which is intended to save/load parameters settings on all tabs of the window 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.
- The **Resources** section, the **Building**, **Filtration**, **Texturing** and **Coord System** tabs;
- The **Use all cores on local PC** checkbox which allows to use all the cores of workstation CPU for calculation operation;



It is necessary to clear the checkbox to use only one core.

- The **Additional parameters** button which allows to set additional parameters of 3D-TIN creation in **Building** and **Texturing** tabs.
2. Select the **data source** in **Resources** section: several selected DEM **files** or **folder** with DEMs.
 - [optional] click the  button to specify the **input data** (DEM **files**);
 - [optional] click the  button to specify the **input data** (a **folder**, containing DEM files).

Click the  button, to specify the **output data** folder;
 3. [optional] Set the following main and additional 3D-TIN **building** parameters in an appropriate tabs (see [Section 5.16](#)).
 4. Click OK to start operation of batch 3D-TIN creation in usual mode.

In order to create 3D-TINs using distributed processing, 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. 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.

If the **Bilateral filter** is used for pre-filtering, when creating a very detailed 3D-TIN surface, it is highly recommended to set **Max MT tasks** for a workstation with a multicore processor and 16 Gb RAM no higher than 1, and to decrease **Tile size** down to 32 (see the **Building** tab),

3. Click the **Distributed processing** button.



Depending on the size of creation area the operation can take a long time.

6. Contour lines

6.1. 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 5. 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	save or rewrite active contours layer
Save as...	to save active layer with new name
Save selected as...	to save <i>just</i> selected contours
Close	to close contour lines layer
Build contours	contains menu items used to create contour lines with help of various source data
Import	contains items used to import contour lines from files with different extensions (see the “ <i>Import of vector objects</i> ” chapter of the “ Vectorization ” User Manual)
Export	contains menu items used to perform export of contour lines to files with different extensions (see the “ <i>Export of vector objects</i> ” chapter of the “ Vectorization ” User Manual)
Check contours intersections...	to check created contours for crossing/self-crossing, that occur after operation of contours smoothing
Check contours by points	to perform quality control of contours creation using regular points, if they were not used for contours creation
Merge contours...	to merge created contours in automatic or manual mode
Check merging contours...	to check contours for breaks that occur after manual or automatic contours merging
Contours connecting	to connect created contours automatically or manually (without merging into a single vector object)
Check contours vertices	to check elevations of created contours vertices
Precision of objects coordinates	to set a precision of vector objects coordinates at level of decimal places

6.2. Contour lines generation

6.2.1. Generation contour lines by TIN

The system provides possibility to create *contour lines* – vector lines, joining points with the same terrain elevation – by [irregular spatial triangles network](#) (TIN).

To create contour lines by TIN, perform the following actions:

1. [Load](#) TIN layer.

2. Choose **Terrain › Contours › Build contours › From TIN...** or use the **Ctrl+N, C** hotkeys. The **Create contours by TIN** window opens.

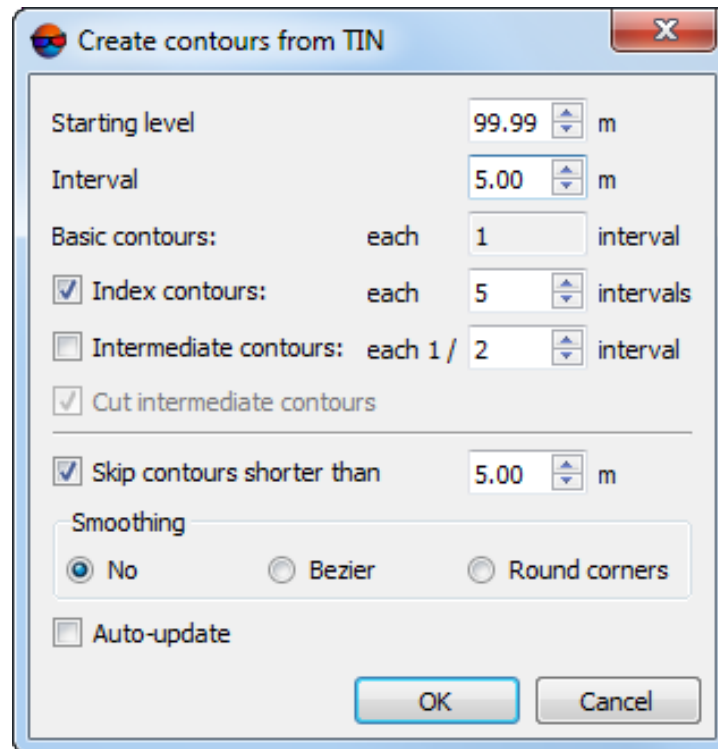


Fig. 85. Parameters of contour lines generation by TIN

3. [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 elevation difference (Zmin, Zmax), right click on the TIN layer in the *Manager* and select the **Information** item, or select **Edit › Active layer › Layer information**. The system displays the information window that contains information about the layer.

4. In the **Interval** field specify contours interval in meters. Interval of **Basic contours** equals to 1 contours **Interval** (displayed in the **Basic contours** field).
5. [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.

6. [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.



The **Cut intermediate contours** checkbox allows to automatically delete fragments of intermediate contours in the areas with increasing slope gradient.

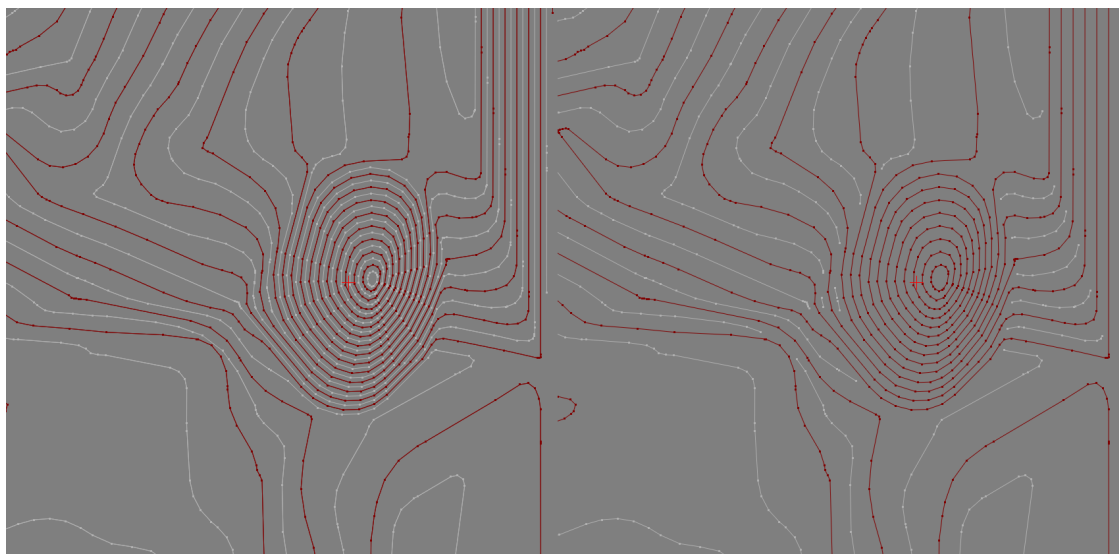


Fig. 86. Deletion of fragments of intermediate contours in the areas with increasing slope gradient

7. [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.
8. [optional] To create contour lines as smooth curves, set the **Bezier** or **Corner rounding** smoothing methods in the **Smoothing** section.
- [optional] if the **Bezier** smoothing method is selected, specify the smoothing degree using the slider.

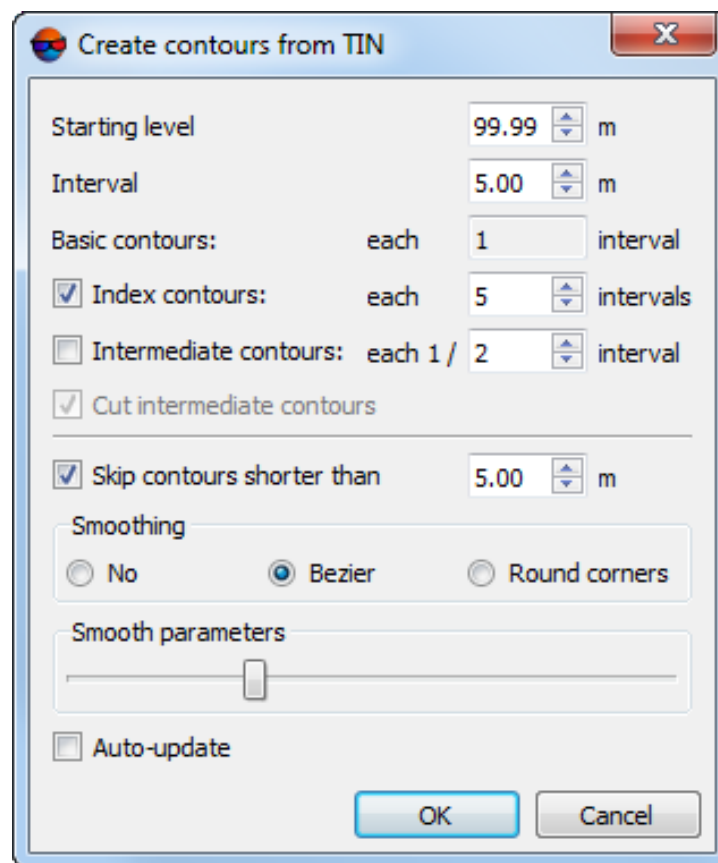



Fig. 87. Parameters of contour lines generation by TIN

- [optional] if the **Corner rounding** smoothing method is selected, input the following smoothing parameters:
 - **Curvature radius** for smoothing curve.
-  Set the **auto** checkbox to calculate **Curvature radius** for smoothing curve automatically.
- **Precision** – maximal distance from polyline segment to a curve in the area between the two closest vertices.

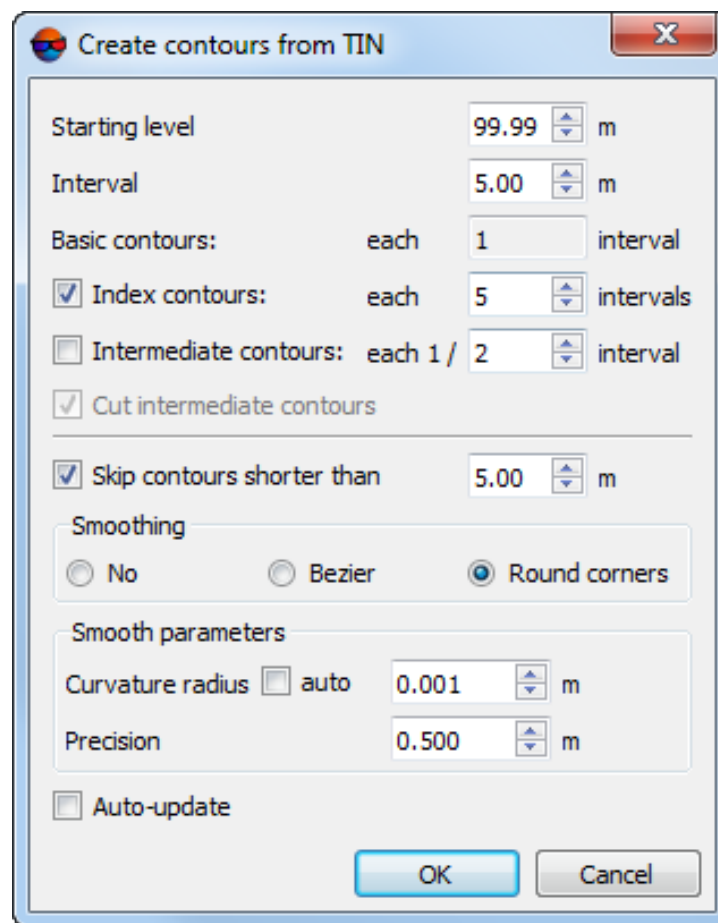


Fig. 88. Parameters of contour lines generation by TIN

9. [optional] Set the **Auto-update** checkbox for automated re-generation of contours created using TIN in case of the TIN layer editing.
10. Click OK. After that the system starts to create contours in a new layer *Contours*. To edit created contours select **Window > Contours classifier table** (see [Section 6.3](#)).

6.2.2. 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....** The **Contours creation parameters** window opens.

The **DEM parameters** section displays information about elevation difference (**Min. elevation**, **Max. elevation**) in DEM.

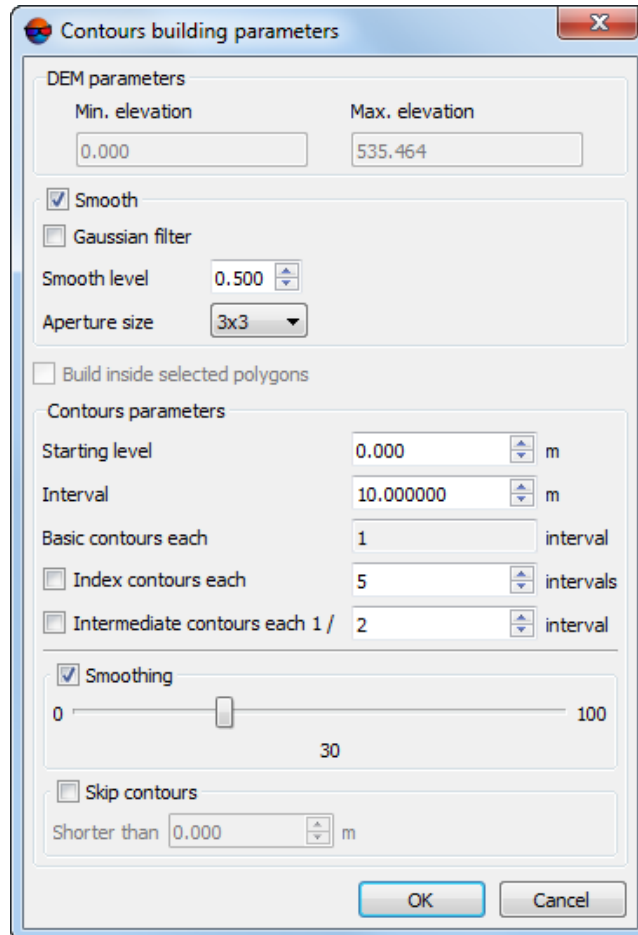


Fig. 89. Parameters of contour lines generation by DEM

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*.

To edit created contours select **Window › Contours classifier table**.

6.2.3. Contours creation in batch mode

The system provides possibility to create contour lines in batch mode. Batch mode allows to create multiple files of contour lines simultaneously and to save them to specified folder.



Contours creation in batch mode is possible in 64-bit system version only.

In order to build contours using TIN in batch mode perform the following actions:

1. Choose **Terrain › Contours › Build contours › In batch mode....** The **Create contours in batch mode** window opens.

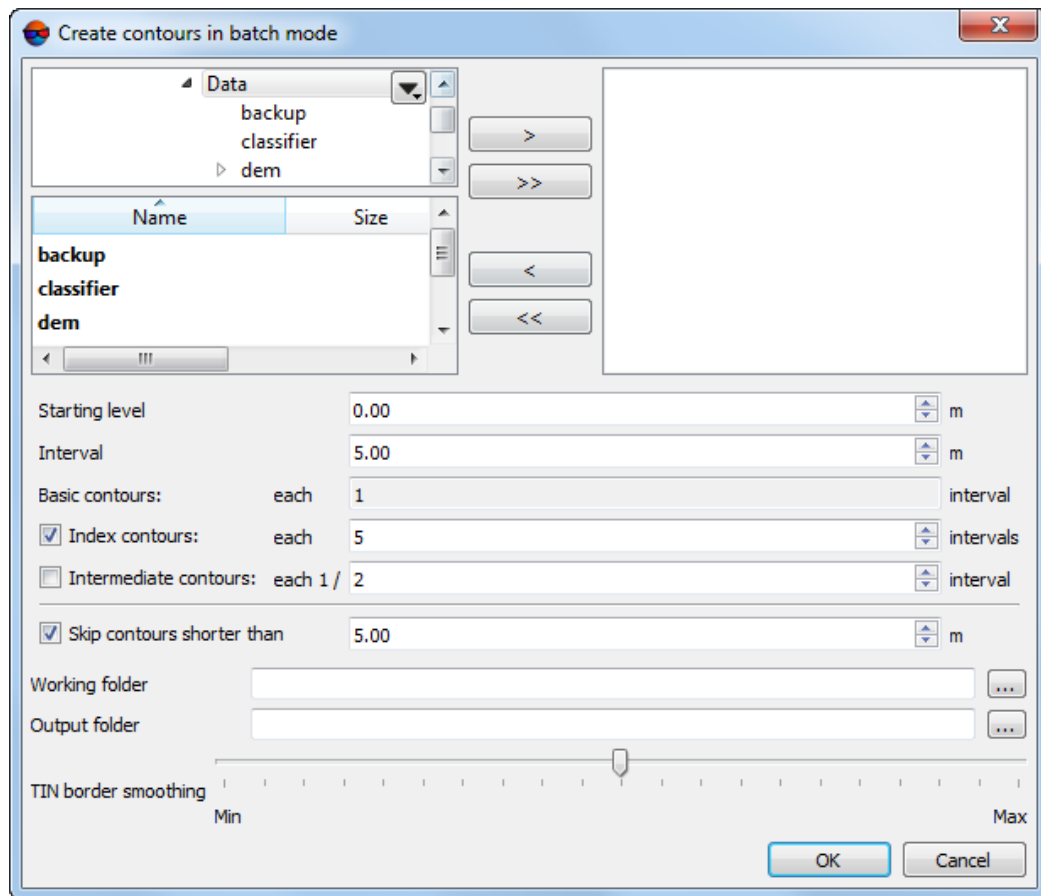












Fig. 90. Parameters of contours creation in batch mode

2. Choose a folder containing vector objects in resources tree.

 The  button allows to display all available resources in nested files. The  button allows to refresh part of the window with resources.


 The  button allows to show a list with 10 recently selected resources.

3. Select file with contours in the list and click the  button to add a layer.

 The  and  buttons allow to add to the list or remove from it all added files with vector objects, the  button allows to remove selected file from the list.

4. Repeat the 2 and 3 steps to add next files with vector objects.

5. Click the  button in the **Working folder** field and select a folder in active profile resources.

6. Click the  button in the **Output folder** field and select a folder in active profile resources to save output contour lines.
7. Configure necessary parameters of contours creation like for **Generation contour lines by TIN**.
8. Click OK to start contour lines creation in batch mode.

To edit created contours select **Window** > **Contours classifier table**.

6.3. Contour lines editing

To edit contours display parameters the system provides **Window** > **Contours classifier table** menu item. Display parameters are saved and applied during creation or loading of new contours layer.

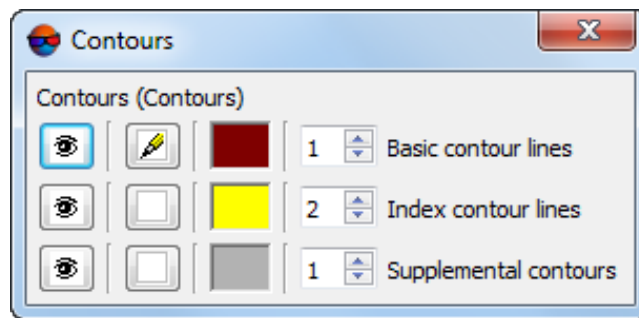




Fig. 91. Contours classifier

The **Contours** window allows to configure the following parameters:

-  – allows to hide/show selected type of contours in 2D-window;
-  – allows to select a type of contours to be edited;
- **Layer color** – allows to configure color to display selected contours type;



To change contour color, double click the **Layer color** button. The **Color** window opens. Choose contour color and click OK.

- input field allows to specify width of selected contours type, that remains the same during scale change.



To change a contour width, specify a value in this input field (this parameter remains the same during scale change).

6.4. Operations with contour lines

Contours are polylines that include straight line segments (broken lines) or arc segments (curves). Contours saving, loading and editing is performed in the same way as for any vector layer (see the “[Vectorization](#)” User Manual).

For contours editing the system provides tools of polylines editing, see description in the “[Vectorization](#)” User Manual.

The system provides for automatic vector object corner round off through the increase of vertex number (see the “Corners round off” and “Convolution smoothing” chapters in “[Vectorization](#)” User Manual).



An angle between the segments is rounded by inscribing a circular arc of a given radius in it.

6.4.1. Saving contour lines

To save and rewrite active contours layer select **Terrain › Contours › Save** or click the layer name by mouse right button in the *Manager* and choose the **Save** in right-click menu.

To save active layer with new name select **Terrain › Contours › Save as...** or click the layer name by mouse right button in the *Manager* and choose the **Save as...** in right-click menu, specify file name and destination in active profile resources.

To save *just* selected contours, select all contours that should be saved and select **Terrain › Contours › Save selected as...** After that vector objects are still associated with classifier.

In order to return to vector layer state at the time of the last save, select **Vectors › Revert**.

The *Contours* layer is opened in the same way as a layer with usual vector objects, see [Section 5.3.2](#).

6.4.2. Export of contours sheets

The system allows to save, edit, and export contours within the given notation sheets to the Panorama format. This is exchange format with *.sit and *.map extensions, that is used in *GIS Map* system.



The system provides the possibility of cutting vector objects of one layer by the boundaries of polygons of another layer (**Vectors › Geometry › Cut objects by selected polygons**). Since both contours and notation sheet boundaries are vector objects, this function can be used to save contours within the selected notation sheets (see the “Cut vectors by selected polygons” chapter of the “[Vectorization](#)” User Manual).

To do this, perform the following:

1. Create notation sheets (see the “Generators of splitting into sheets” chapter of the “[Vectorization](#)” User Manual);
2. Select notation sheets within which you need to save/export contours;
3. Choose **Terrain > Contours > Prepare/export contours sheets**. The **Prepare/export contours sheets** window opens:

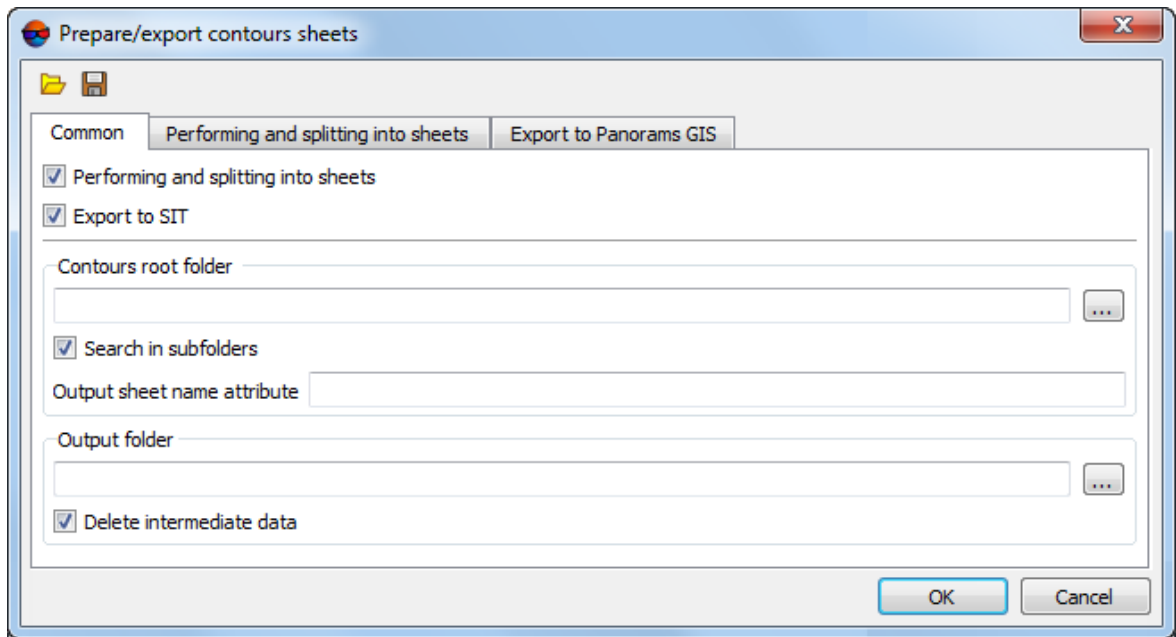




Fig. 92. The Prepare/export contours sheets window

4. In the **Common** tab set the following options:
 - [optional] Clear the **Performing and splitting into sheets** checkbox in order not to save contours within the selected notation sheets (see the **Performing and splitting into sheets** tab);
 - [optional] Clear the **Export to SIT** checkbox in order not to export contours within the selected notation sheets to the Panorama format (see **Export to Panorama GIS** tab);
 - In the **Contours root folder** field click the  button and select the folder containing initial contours in the active profile resources;
 - [optional] Clear the **Search in subfolders** checkbox in order not to search initial contours in subfolders;

- In the **Output sheet name attribute** field, input the attribute's name where notation sheet names are recorded;



A name of the attribute where notation sheet names are recorded (Name, by default) is set during notation sheet creation (see the “Generators of splitting into sheets” of the “[Vectorization](#)” User Manual). To check the attribute's name, select a notation sheet and choose **Window > Object attributes**.

- In the **Output folder** field input the path to save contours or click the  button to choose the folder name and the path in the active profile resources to save contours within the selected notation sheets;
- [optional] Clear the **Delete intermediate data** checkbox in order not to delete processing data after calculation accomplishment.

5. In the **Performing and splitting into sheets** tab, set the options for contours editing within the selected notation sheets:



To save contours within the selected notation sheets (without additional editing), clear all the checkboxes in the **Performing and splitting into sheets** tab.

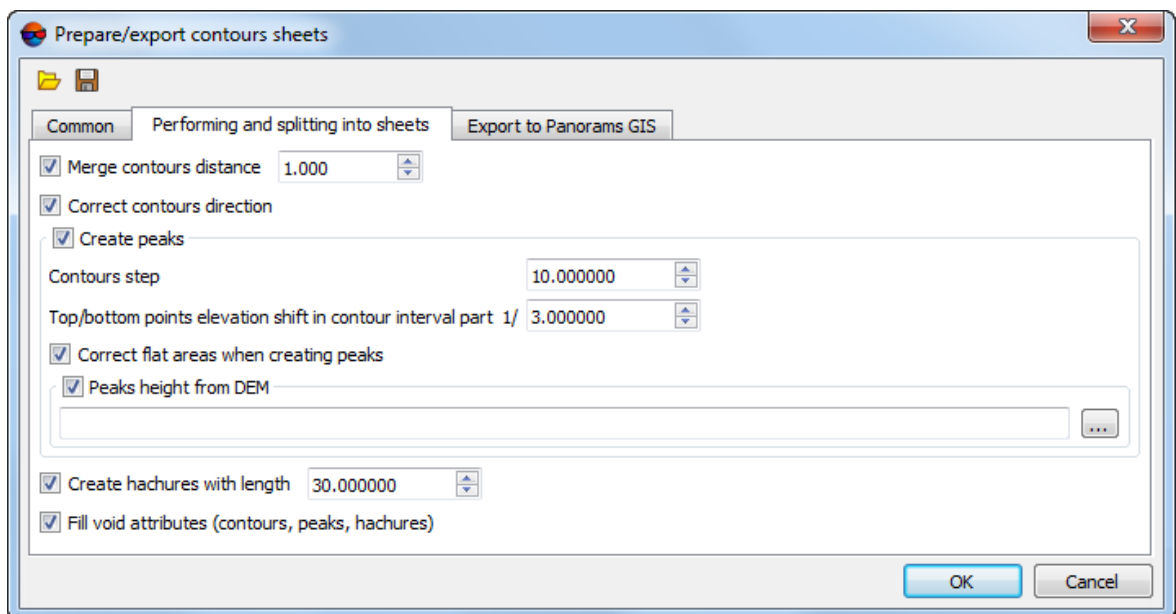


Fig. 93. The Processing and splitting into sheets tab

- [optional] For automated [merging](#) of contours with “breaks”, the default checkbox **Merge contours distance** is set. Clear the checkbox not to merge contours, or input maximal distance between contours in meters, at which the system performs contour merging.
- [optional] In order not to **Correct contours direction**, clear the appropriate checkbox (only for the Panorama format);

- [optional] To avoid adding point vector objects denoting peaks and pits, clear the **Create peaks** checkbox;
- Input the **Contours step** in the appropriate field;




To get this information from the project, open the layer with contours.

- To set up the parameters to calculate elevations of point objects denoting peaks and pits, set **Top/bottom points elevation shift in contour interval part**;



For example, if **Top/bottom points elevation shift in contour interval part** is 1/3, then the elevation of a point denoting an upland point will be a sum of the value of the elevation of the closest contour to the point and one third of contour spacing. **Precision of peak points** is set in the **Export to Panorama GIS** tab (see below).

The system also allows to clarify elevations of point objects denoting peaks and pits using a DEM together with **Contours step** and **Top/bottom points elevation shift in contour interval part** parameters (see below).

- [optional] In order not to **Correct flat areas when creating peaks**, clear the appropriate checkbox;
- [optional] To clarify elevations of point objects denoting peaks and pits, the **Peaks height from DEM** checkbox is set by default. To avoid taking DEM into account when calculating elevations of point objects denoting peaks and pits, clear this checkbox or click the  button to select DEM in the active profile resources;
- [optional] To create hachures when saving contours within selected notation sheets, the default **Create hachures with length** checkbox is set. Clear this checkbox in order not to create hachures or input hachures length, in meters;
- [optional] Clear the **Fill void attributes (contours, peaks, hachures)** checkbox, if this action is not required;



Filling void attributes can be required, for example, for additional vector objects (contours) created manually.

6. In **Export to Panorama GIS** tab, set options for contour's export to Panorama exchange format within the selected notation sheets:

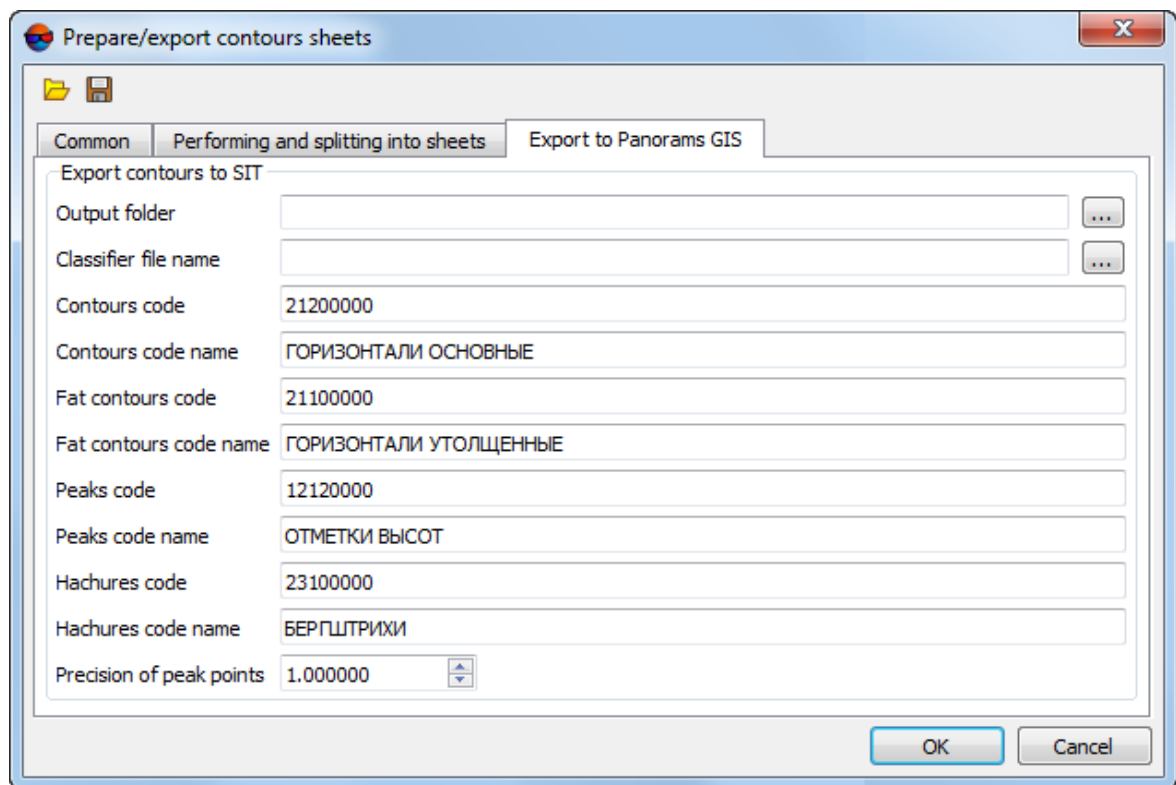




Fig. 94. The Export to Panorama GIS tab

- In the **Output folder** field, enter the path to save exchange files or click the  button to choose the path for files saving in the file system;
- To indicate the path to the file with classifier, click the  button in the **Classifier file name** field;
- [optional] Edit **Contours code**, if needed;
- [optional] Edit **Contours code name**, if needed;
- [optional] Edit **Fat contours code**, if needed;
- [optional] Edit **Fat contours code name**, if needed;
- [optional] Edit **Peaks code**, if needed;
- [optional] Edit **Peaks code name**, if needed;
- [optional] Edit **Hachures code**, if needed;
- [optional] Edit **Hachures code name**, if needed;
- Set the **Precision of peak points**.

7. Click OK.

Contours located within selected notation sheets are saved in the preset output folder in the active profile resources in separate folders whose names correspond to notation sheet names.

Files from the Panorama exchange format are saved in the selected output folder in the file system.

6.4.3. Contours merging

The system provides possibility to merge created contours in automatic or manual mode.

In order to merge contours **manually**, perform the following actions:

1. [Load](#) or [create](#) a layer with contour lines.
2. Select **Terrain › Contours › Load** and open a file containing contour lines from adjacent stereopair. The **Loading** window opens.
3. Click the **Add new data to ... layer without unloading existing data** button to add contours from adjacent stereopair to a source layer.
4. Use tools intended for polylines editing (as well as snapping) for contours manual merging (see the “Snapping mode” and “The ‘Topology’ menu” chapters of the [Vectorization](#)” User Manual).

In order to start automatic contours merging, perform the following actions:

1. Select **Terrain › Contours › Merge contours....** The **Parameters** window on the **Source contours** tab opens.

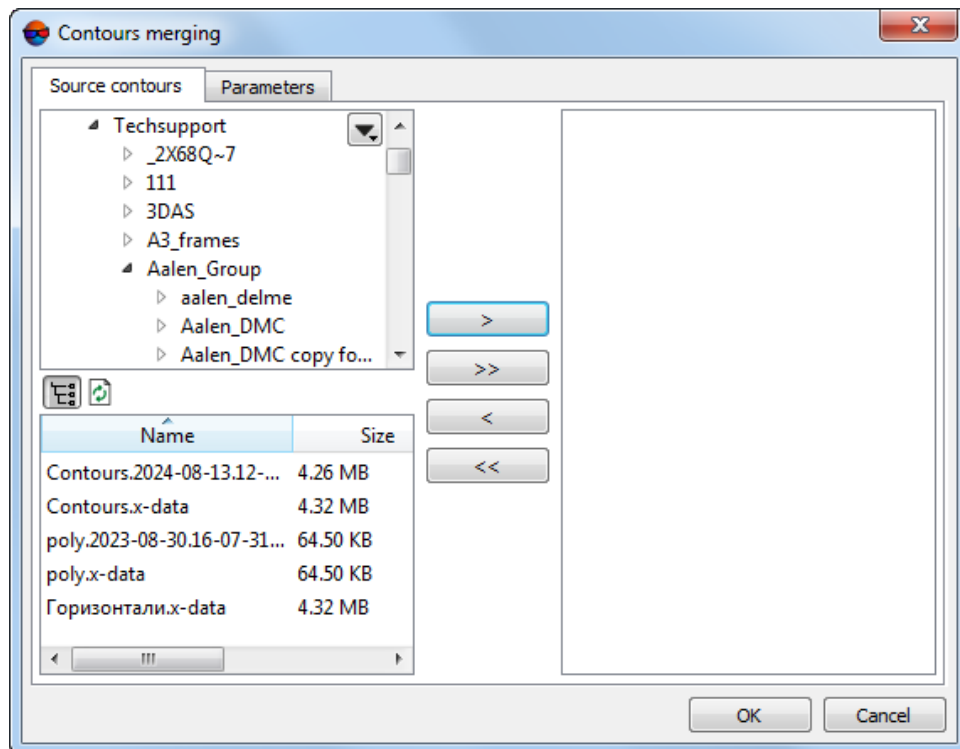


Fig. 95. Parameters of contours merging on the 'Source contours' tab

2. Choose a folder containing contours in resources tree.

The button allows to display all available resources in nested files. The button allows to refresh part of the window with resources.

The button allows to show a list with 10 recently selected resources.

3. Select file with contours in the list and click the button to add a layer to be merged.

The and buttons allow to add to the list or remove from it all added files with contours, the button allows to remove selected file with contours from the list.

4. Repeat the 2 and 3 actions to add next files with contours.



It is necessary to choose for merging at least two files containing contour lines.

5. Click the **Parameters** tab.

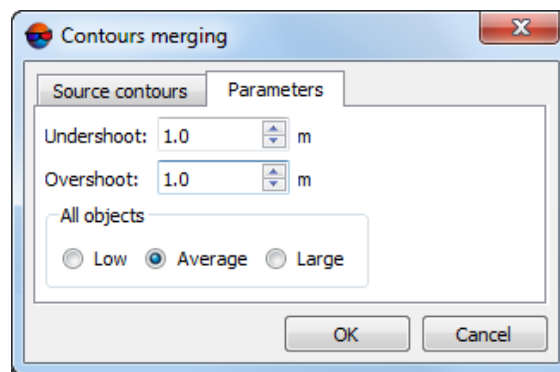


Fig. 96. Parameters of contours merging on the 'Parameters' tab

6. [optional] Input maximal distance between contours in meters, at which the system performs contours merging, to the **Undershoot** field.
7. [optional] To the **Overshoot** input acceptable part of contour line to be deleted in meters.
8. In the **Smooth level** section specify a smoothing level of smoothed contour from vertices of unsmoothed one: **Low**, **Average** or **Large**.
9. Click OK. New vector layer that contains merged contours is created in the *Manager*.



The system also allows to [connect](#) contours (without merging contours into a single vector object and creating a new layer).

6.4.4. Check of contours merging quality

The system provides possibility to check contours for breaks that occur after manual or automatic contours merging.

To perform check of contours merging, select **Terrain > Contours > Check merging contours** The **Parameters** window opens.

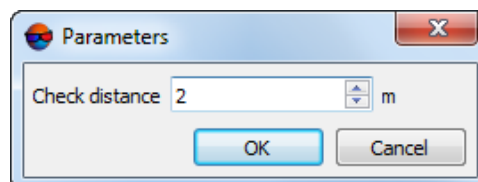


Fig. 97. The Parameters window

In the **Check distance** input field specify maximal distance between contours in meters, when the system starts to search for contours breaks, and click OK.

If the contours breaks are detected the **Erroneous contours vertices** window opens.

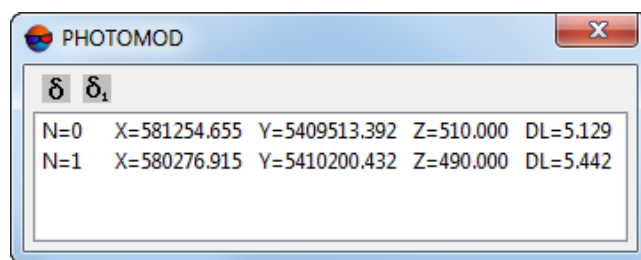


Fig. 98. Erroneous contours vertices

The window displays a list of contour lines breaks, where:

- N – is break serial number;
- X, Y, Z – coordinates of break's middle point;
- DL – distance between contours in meters.

Select a line in the list by mouse click. After that marker is placed to the break's middle point and is displayed in 2D-window.

To 'connect' contour break, select the contour in the list and click the δ_1 button.

To 'connect' all found contour breaks, click the δ button.

6.4.5. Contours connecting

The system provides possibility to connect created contours (without [merging](#)) in automatic or manual mode.

In order to connect contours **manually**, perform the following actions:

1. [Load](#) or [create](#) a layer with contour lines.
2. Select **Terrain > Contours > Load** and open a file containing contour lines from adjacent stereopair. The **Loading** window opens.
3. Click the **Add new data to ... layer without unloading existing data** button to add contours from adjacent stereopair to a source layer.
4. Use tools intended for vertices editing (as well as snapping) for contours manual connecting (see the "Snapping mode" and "Vector objects editing" chapters of the [Vectorization](#) User Manual).

In order to start automatic contours connecting, perform the following actions:

1. Select **Terrain > Contours > Contours connecting....** The **Contours connecting** window opens.

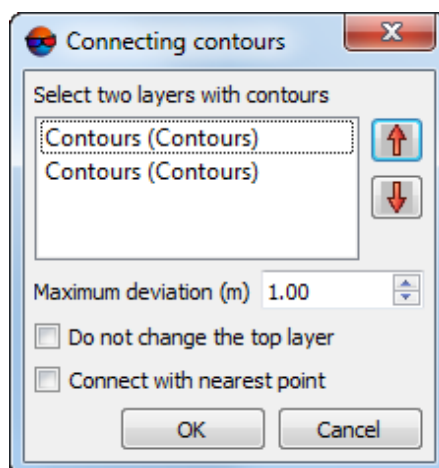


Fig. 99. The Contours connecting window

2. In the list of opened vector layers, select two layers with contours;
3. Input maximal distance between contours in meters, at which the system performs contours connecting, to the **Maximum deviation** field.
4. [optional] Set the **Do not change the top layer** checkbox to move contour vertices only within one of the selected layers (otherwise vertices will be moved within both selected layers during the connection operation);



The ↑ and ↓ buttons allows to change positions of the selected layer in the list of downloaded vector layers.

5. Click OK. Ending contour vertices belonging to different layers, the distance between which does not exceed **Maximum deviation**, converge (in central position in the plan if the **Do not change the top layer** checkbox is cleared), otherwise ending vertices from the lower layer will be moved to positions of vertices belonging to the upper layer.

If there will be 3 or more vertices for “connecting” contours having one elevation at a distance not exceeding **Maximum deviation**, the **Unconnected points** window opens. The window contains the table with vertices which may require **manual** editing. If a line in the table is selected, the marker is moved to the appropriate vertex.

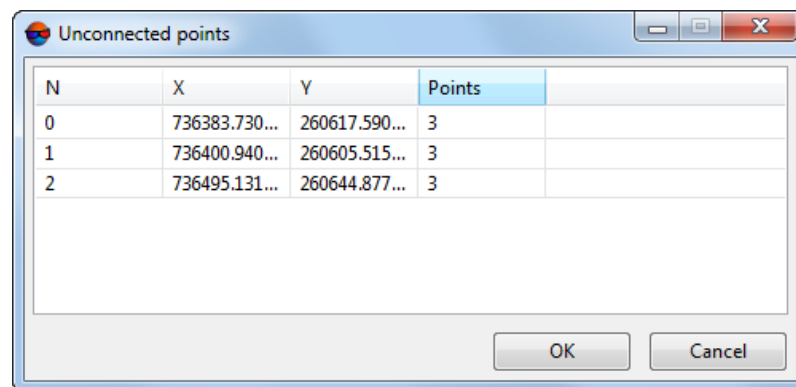


Fig. 100. The Unconnected points window



The system also provides for [merging](#) contours (a new layer containing contours merged into single vector objects is created).

6.4.6. Precision of objects coordinates

The system allows to set a precision of vector objects coordinates at level of decimal places.

A need to set decimal places for contour heights can be arisen after recalculation of vector objects coordinates from one coordinate system to another one (see the “Transformation of objects coordinates” chapter of “[Vectorization](#)” User Manual).

To set a precision of vector objects coordinates, perform the following:

1. Create or load vector layer;
2. [optional] Select vector objects;
3. Select **Terrain > Contours > Precision of objects coordinates**. The **Objects precision** window opens;

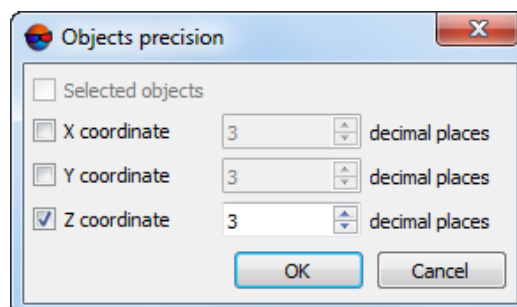


Fig. 101. The “Objects precision” opens

4. [optional] Turn off **Selected objects** checkbox to apply parameters of coordinates precision for the all objects of vector layer;
5. Set number of **decimal places** for **X, Y, Z** coordinates;



If **decimal places** are zero coordinates of vector objects will be rounded to integers.



If **decimal places** are negative numbers coordinates of vector objects will be rounded to points (i.e. "-1" value leads to rounding to units, "-2" value leads to rounding to tens etc.).

6. Click OK. As a result, selected or all the objects of vector layer are recalculated according to specified precision of coordinates.

6.5. Accuracy control of contours creation

6.5.1. Control of contours crossing

The system allows to check created contours for crossing/self-crossing, that occur after operation of contours smoothing.

In order to check contours for crossings perform the following actions:

1. [Load](#) or [create](#) a layer with contour lines.
2. Select **Terrain > Contours > Check contours intersections....** The **Question** window opens.

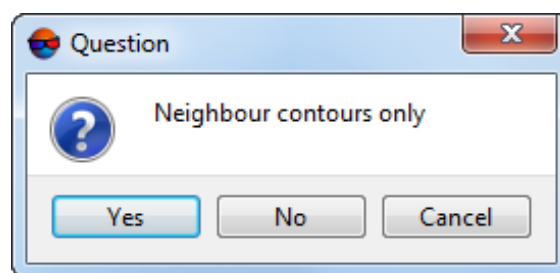


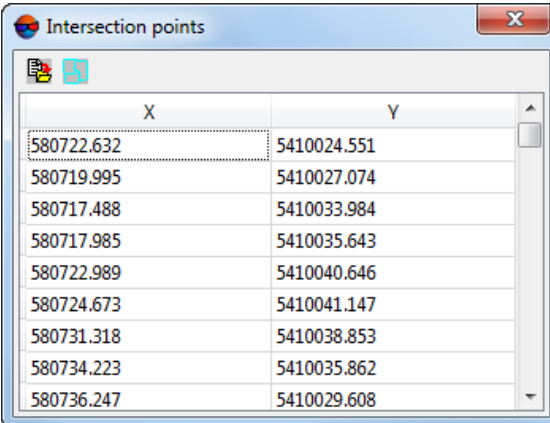
Fig. 102. Parameters of contours intersections control

3. [optional] Click **Yes** to check intersections of **neighbour contours only**. Click **No** to check all contour lines for intersections.




It may take a long time to check all contour lines for intersections.

4. Click OK. The system start operation of created contours check for intersections. When the check is completed the system opens the **Intersection points** window.



X	Y
580722.632	5410024.551
580719.995	5410027.074
580717.488	5410033.984
580717.985	5410035.643
580722.989	5410040.646
580724.673	5410041.147
580731.318	5410038.853
580734.223	5410035.862
580736.247	5410029.608

Fig. 103. Coordinates of intersection points of contour lines

In order to export a list of contours intersection points to file with csv extension, click the  button.

To create a new layer that contains intersection points as points, click the  button.

5. Select a point in the list by mouse click. After that marker is placed to the crossing point and is displayed in 2D-window.

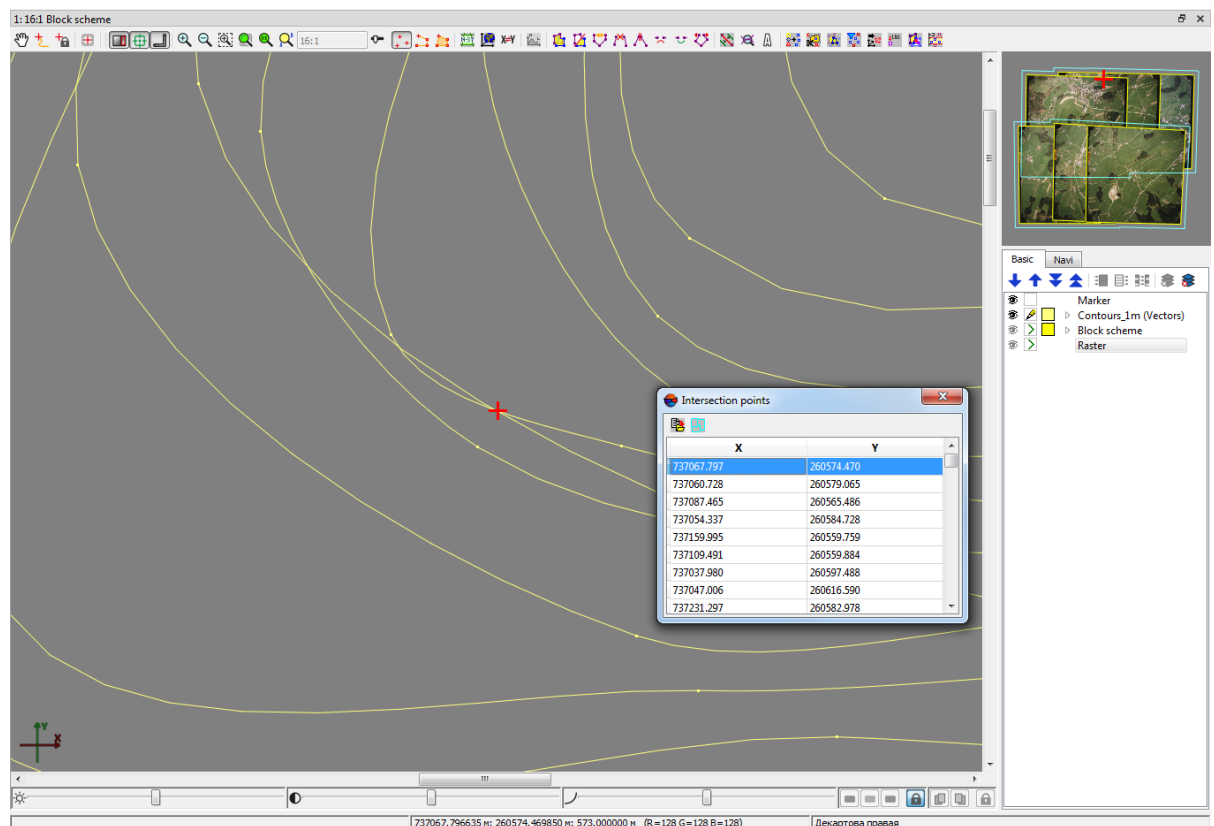


Fig. 104. Control of contours crossing

6. Use tools of polylines editing to edit contours and correction of intersection\self-intersection errors (see description in the [Vectorization](#) User Manual).

6.5.2. Check contours by points

The system provides opportunity to perform quality control of contours creation using regular points, if they were not used for contours creation.



To start the check operation it is necessary that the *Contours* layer and the *Vectors* layer (containing points) are loaded and displayed in the *Manager*.

In order to start contours check operation by points, perform the following actions:

1. Load or [create](#) a layer with points.
2. Make active the layer with contour lines.
3. Select **Terrain > Contours > Check contours by points**. The **Parameters** window opens.

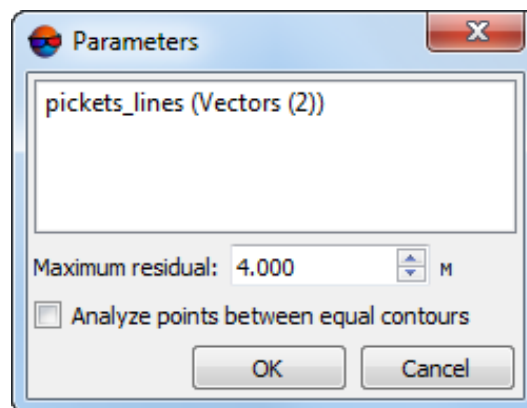
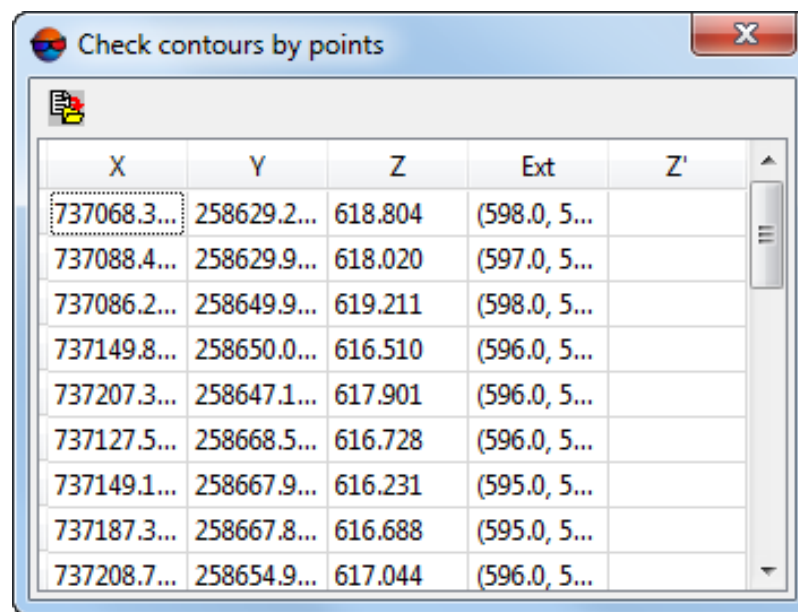


Fig. 105. Parameters of contours check using points

4. Select a layer containing points.
5. In the **Maximum residual** field input a value of maximal deviation of contours from a value of points elevation by Z (in meters).
6. [optional] In order to include to check operation points located on contours with the same elevation (for example, N=27 (Ext=597.0, 597.0) and N=28 (Ext=597.0, 597.0)), set the **Analyze points between equal contours** checkbox on.
7. Click OK. The **Check contours by points** window opens.




The screenshot shows a window titled "Check contours by points" with a close button (X) in the top right corner. Inside the window is a table with five columns: X, Y, Z, Ext, and Z'. The table contains ten rows of data. The first row is highlighted with a dashed border. To the left of the table is a small icon of a document with a red flag. To the right of the table is a vertical scrollbar.

X	Y	Z	Ext	Z'
737068.3...	258629.2...	618.804	(598.0, 5...	
737088.4...	258629.9...	618.020	(597.0, 5...	
737086.2...	258649.9...	619.211	(598.0, 5...	
737149.8...	258650.0...	616.510	(596.0, 5...	
737207.3...	258647.1...	617.901	(596.0, 5...	
737127.5...	258668.5...	616.728	(596.0, 5...	
737149.1...	258667.9...	616.231	(595.0, 5...	
737187.3...	258667.8...	616.688	(595.0, 5...	
737208.7...	258654.9...	617.044	(596.0, 5...	

Fig. 106. Points coordinates during control of contours using points

The window displays a list, where:

- N – is point serial number;
- X,Y,Z – point coordinates;
- Ext – distance between contours in XY plane;
- Z' – elevation difference between contours and points, that exceeds maximal deviation.

In order to export the list to file with *.csv extension, click the  button.

8. Select a point in the list by mouse click. After that marker is placed to the selected point and is displayed in 2D-window.

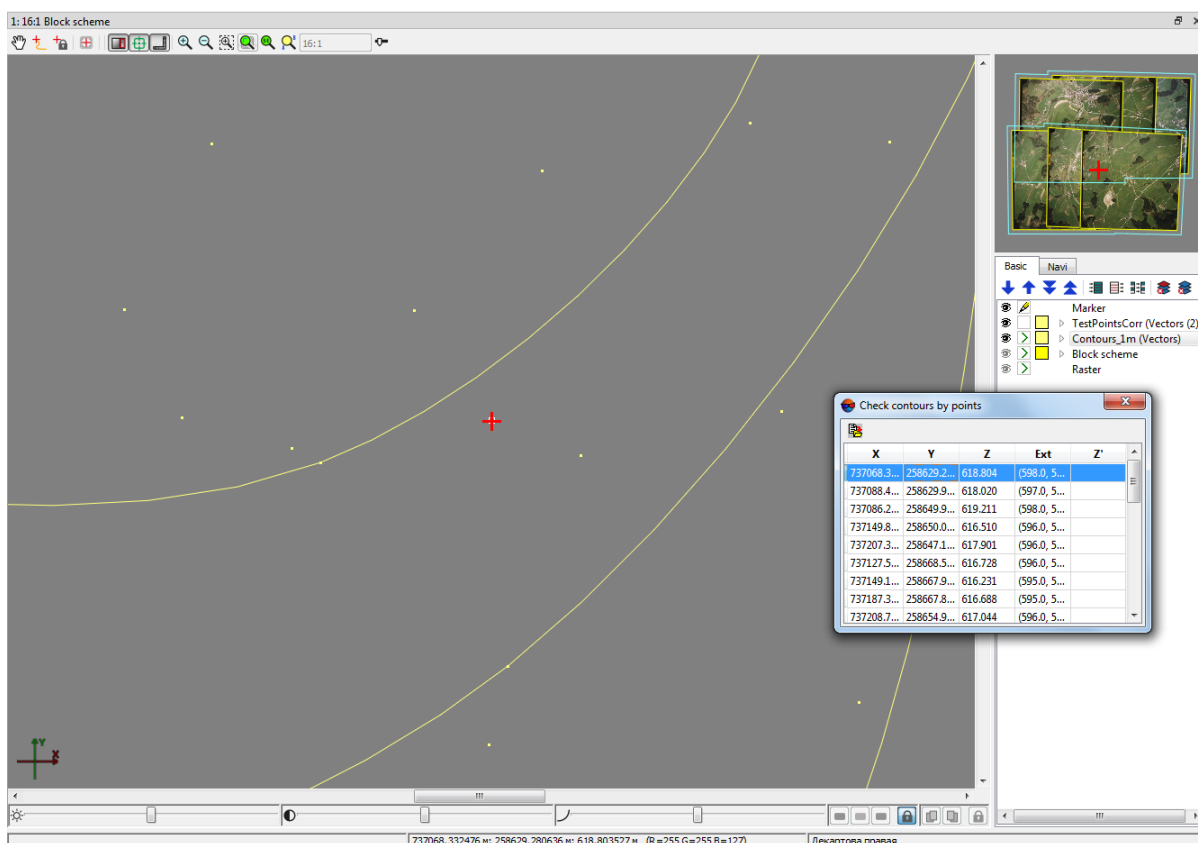


Fig. 107. Check contours by points

9. Use tools of polylines editing to edit contours (see description in the “[Vectorization](#)” User Manual).

6.5.3. Checking elevations of contour vertices

The system provides for the possibility of checking elevations of contours' vertices, as well as whether contour elevations correspond to estimated contour interval. This function may be required if the contours were created by the user manually.

For this, perform the following:

1. [Load](#) or [create](#) a layer with contours
2. Choose **Terrain > Contours > Check contour vertices**. The **Check contour elevations** window opens.

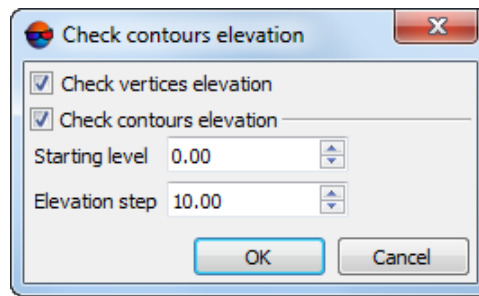


Fig. 108. The Check contour elevations window

- To **check vertex elevations**, set the appropriate checkbox. The system checks whether all contour vertices have the same elevation;
- To **check contours elevation**, set the appropriate checkbox. To successfully complete this operation, it is necessary to correctly set the **start level** of contours and their **elevation step** (the user must do these in advance). As a result, a check is made whether all the elevations of the contours increase relative to the **start level**, in strict adherence to the specified **elevation step**.



Simultaneous check of vertices elevations and contour elevation is not recommended.

3. Click OK.

The contour check runs. If there are no errors, an appropriate message is displayed:

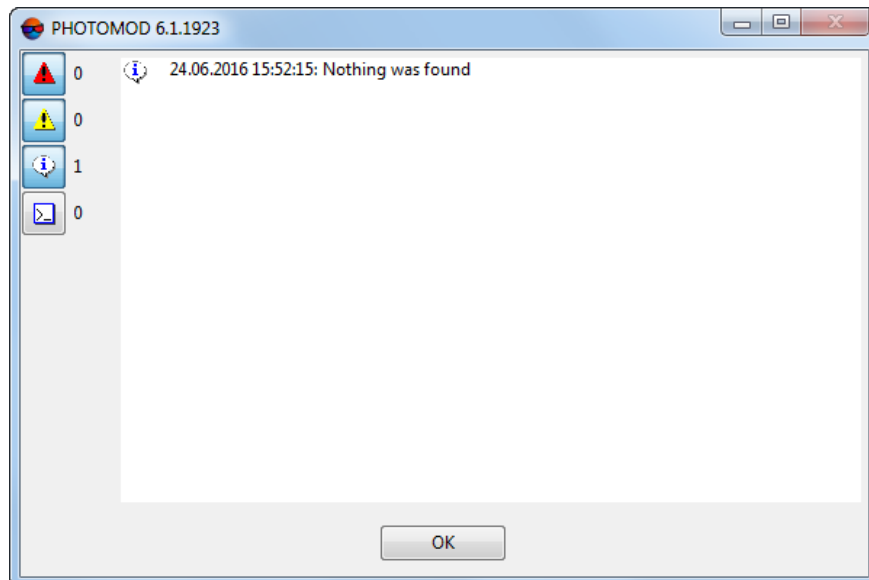
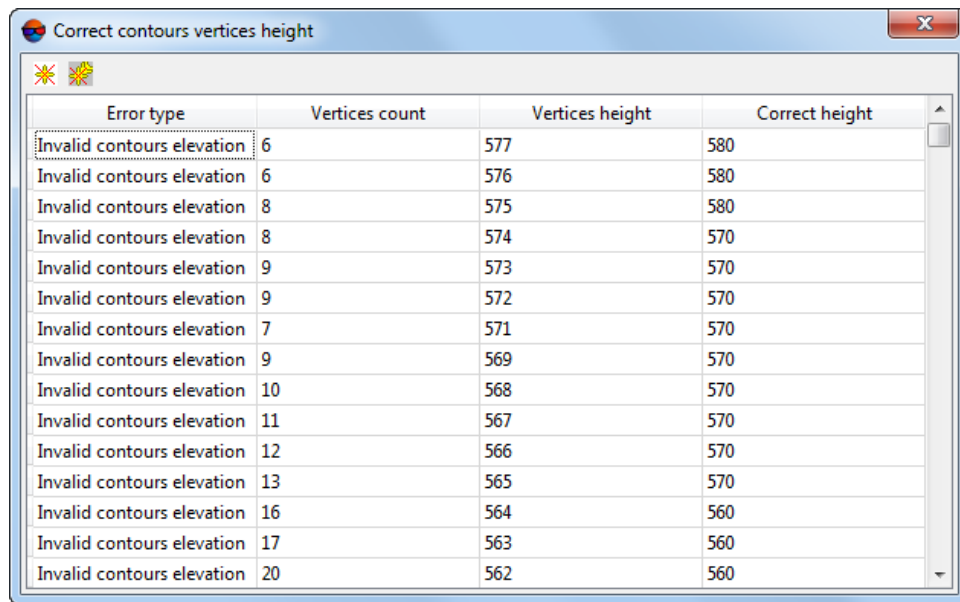


Fig. 109. Informational message about no errors

In case of errors, the **Correct contour vertices height** window opens:



Error type	Vertices count	Vertices height	Correct height
Invalid contours elevation	6	577	580
Invalid contours elevation	6	576	580
Invalid contours elevation	8	575	580
Invalid contours elevation	8	574	570
Invalid contours elevation	9	573	570
Invalid contours elevation	9	572	570
Invalid contours elevation	7	571	570
Invalid contours elevation	9	569	570
Invalid contours elevation	10	568	570
Invalid contours elevation	11	567	570
Invalid contours elevation	12	566	570
Invalid contours elevation	13	565	570
Invalid contours elevation	16	564	560
Invalid contours elevation	17	563	560
Invalid contours elevation	20	562	560

Fig. 110. The Correct contour vertices height window

The window displays a table. Each of the lines contains the following information:

- Number of the error entry;
- Type of error detected and number of involved vertices:
 - **Invalid vertices elevations** and total number of vertices with an error;
 - **Invalid contour elevation** and total number of contour vertices.



A large number of such errors may indicate that the **elevation step** was ill-specified by the user

- Current **vertices height**;
- Estimated **correct height**.

If for some reason it is impossible to determine the correct value of elevation of a particular contour, the entry ??? is displayed in the respective line instead of this value.




Such errors, especially numerous, may indicate that the data being checked are initially incorrect (for example, the checked vector objects are not contours).

4. The system provides for automated correction of contour vertex elevations.



It is recommended to correct sporadic **Invalid vertices elevations** in this way. If an **Invalid contour elevation** was detected, it is recommended to first find out the cause of this error.

- [optional] To **correct selected contour**, select the appropriate error entry and click ;
- [optional] If the **correct height** is not known, the **Object vertices height** window opens:

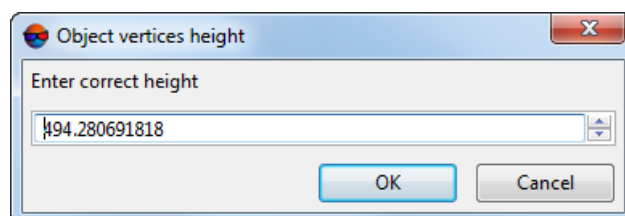


Fig. 111. The Object vertices height window

Enter correct height and click OK.

- [Optional] To **correct all contours**, click .



It is strongly discouraged to **correct all contours** if the **correct height** is unknown for some objects.

7. Digital Elevation Model

7.1. 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 6. Brief description of the DEM menu

Menu items	Function
Load DEM... (Ctrl+O, D)	to load DEM from *.x-dem file
Open from file...	to load DEM from external data without its conversion to internal format
Recent	to perform the quick access to recently loaded DEM files
Save copy	to save opened DEM to a new file
Save selection	to save DEM area
Save as georeferenced raster...	to save a DEM as a raster file with geodetic reference
Close	to close DEM
Close all opened layers	to close all opened DEM layers

Menu items	Function
Layers visibility	contains menu items which allow to perform batch management of DEM layers visibility in the <i>Layer manager</i>
Build DEM...	contains menu items used to create DEM with help of various source data
Rebuild by TIN	to recreate DEM after changing of TIN base layer
Restore	to restoring of DEM consistency , which may be disrupted by applying various operations or DEM filtering
Filter	contains menu items which can be used to filter DEMs
Correct along vector line	to correct DEM cell values along the selected linear objects
Fill null cells	contains menu items used to restore blank cells of DEM using various methods
Cut by selected polygons	to edit coverage area of DEM
Cut null edges	to remove edge areas of DEM, that include blank cells
Transpose	to transpose DEM from the left coordinate system to the right one and vice versa
Transform to another coordinate system	to change coordinate system of DEM
Batch transform to another coordinate system	to change coordinate system of multiple DEMs
Rebuild considering last adjustment	to recreate DEM considering results of the last adjustment (if the project was adjusted after DEM creation once more)
Shift	to parallel displacement of DEM by axes
Split DEM into sheets	to split DEM into sheet to save DEM in parts in separate files
Merge DEMs	to merge DEMs , that have overlap area
Accuracy control	to perform accuracy control DEM creation using various data
Calculate slope map	to calculate a slope map
Convert to LAS	to convert DEM to point cloud (LAS)
Convert to points...	to convert DEM cells to points with specified simplifying
Write points to DEM	to add points to DEM that helps to refine DEM cells values
Export...	contains menu items used to perform export of DEM to different formats
Import...	contains items used to import DEM from files with *.grd, *.asc, *.tif, *.dem, *.mtw, *.dt1, *.dt2, *.img, *.pix, *.hgt extensions

7.2. General information

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).

DEM is displayed in 2D-window, and also could be opened to view in 3D-window or in 2D-window of stereopair in stereo mode.



DEM in stereo mode may be displayed inaccurately.

To view information about DEM select **Information** item in the *Manager*. The system displays information window which includes the following information about the layer:

- layer name;
- full path to layer file;
- resource size;
- file creation time;
- presence of data and changes in the layer;
- possibility to change the layer;
- coordinates of DEM border nodes, including Z-coordinate;
- ground sample distance;
- DEM size (number of cells);
- bytes per sample;
- layer coordinate system.

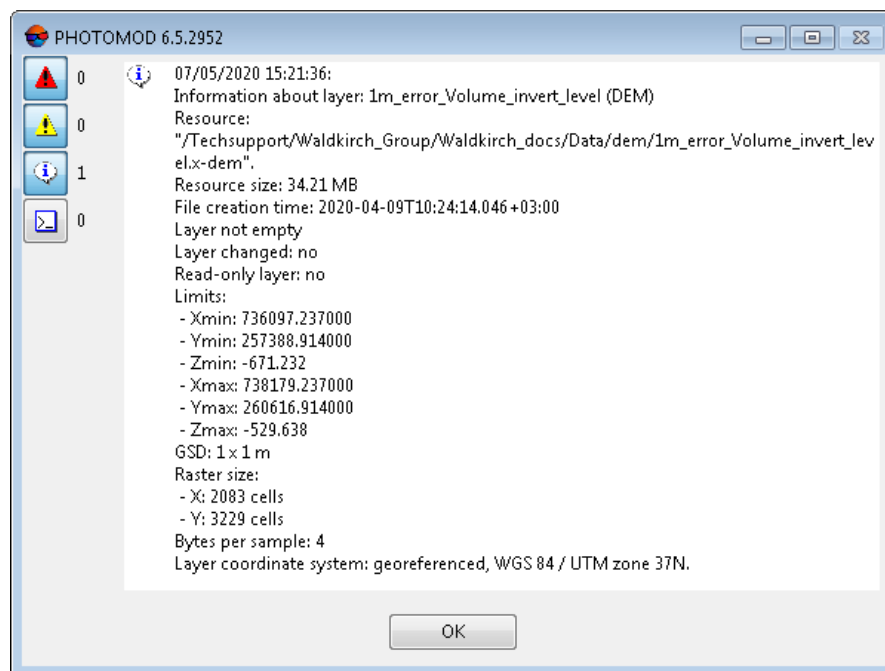


Fig. 112. Information about DEM layer

7.2.1. DEM layer display settings

For *DEM* layers could be changed *transparency* of layer visibility in 2D-window.



Changing of layer transparency is possible only if the **Do not use textures for precision raster display** checkbox is set off on a **System** tab of the **Settings** window (see the [General systems parameters](#) User Manual).

To do this perform the following:

1. Double-click on layer objects color button () in a *Layer manager*. The **Layer display settings** window opens.

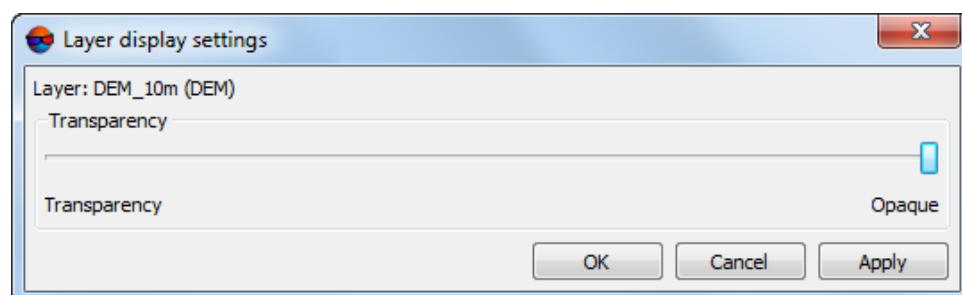


Fig. 113. Layer display settings

2. Set the layer **transparency**.

3. Click the **Apply** button to display changes in 2D-window.
4. Click OK to save parameters.

7.3. DEM creation

7.3.1. 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)**. The **DEM by TIN creation parameters** window opens.

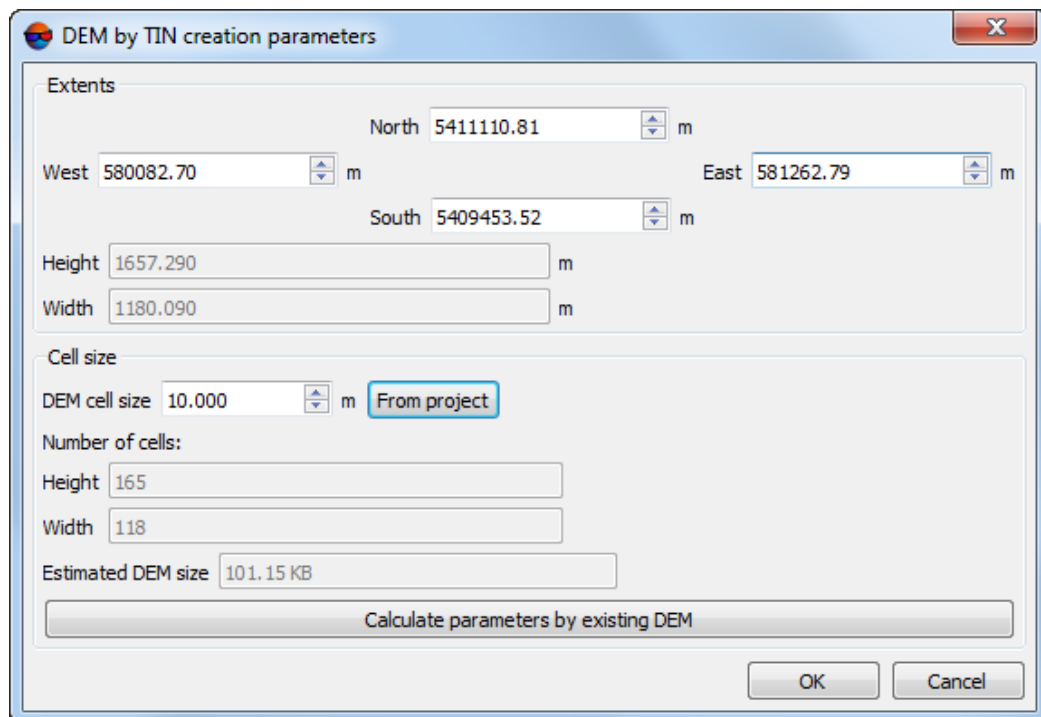


Fig. 114. DEM by TIN creation parameters

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** (see also [Section 7.10](#) and [DEM load](#)). 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.

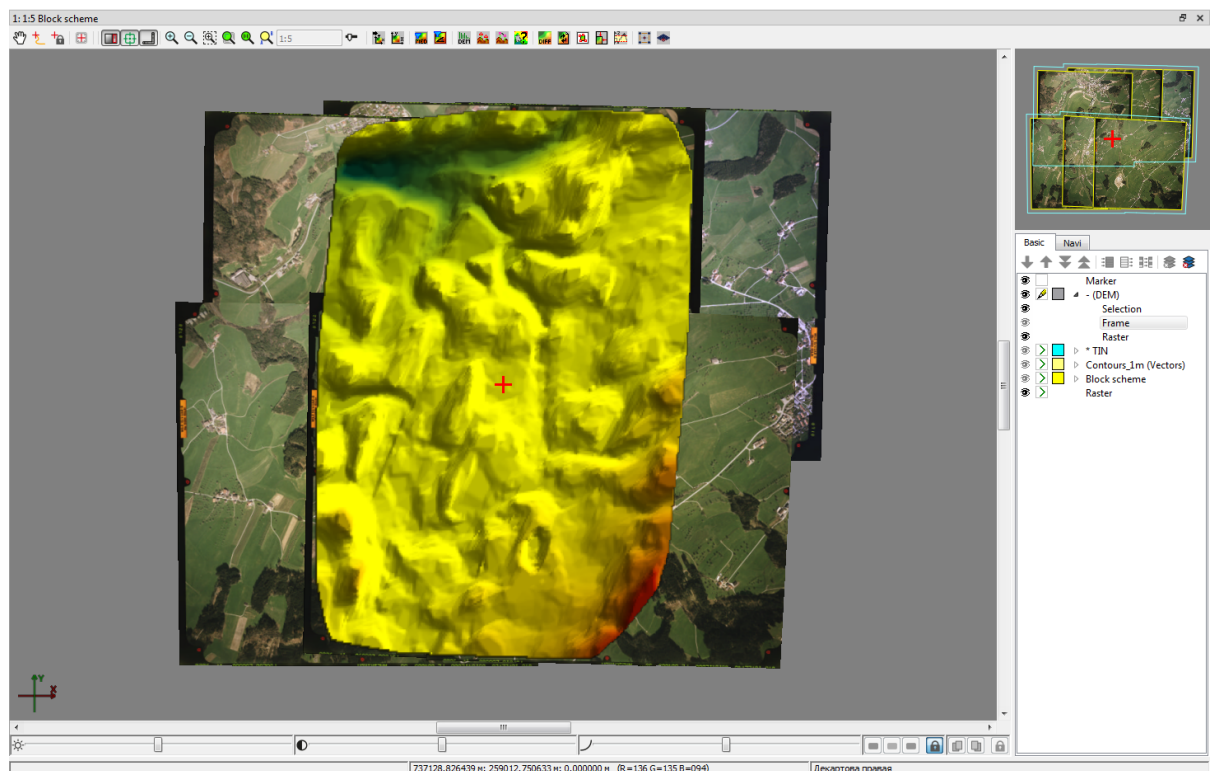


Fig. 115. DEM built from TIN

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.

7.3.2. DEM creation using regular points

The system provides possibility to create DEM using a set of points created in regular grid nodes or imported from external files. Perform the following actions to do this:

1. [Create](#) or load a layer with points.
2. Select **Terrain › DEM › Build DEM › From regular points...** The **Parameters of DEM building by regular points** parameters window opens.

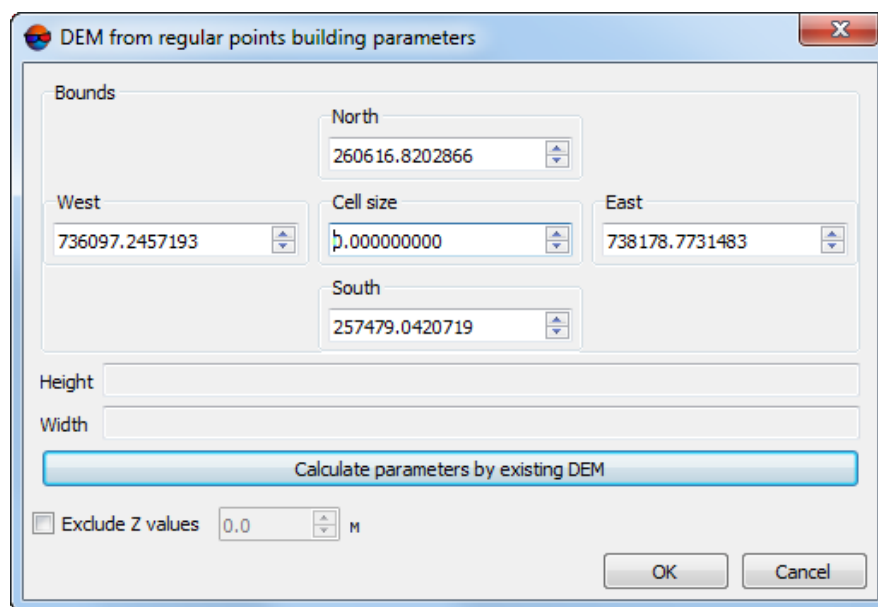


Fig. 116. DEM creation using regular points

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, as well as number of DEM nodes.



Coordinates of border points of source layer are default values.

4. Specify **Cell size** in meters to change size of output DEM cell.



DEM cell size should be commensurate with the average distance between points. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

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. [optional] In order to exclude points with specified elevation from creation operation, set the **Exclude Z values** checkbox on and input the parameters in meters.
7. Click OK. Specify DEM file name, define a folder in active profile resources and click **Save** (see also [Section 7.10](#) and [DEM load](#)). 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.

7.3.3. DEM creation using points

The system provides possibility to create DEM using points (point vector objects located on the terrain relief surface). Perform the following actions to do this:

1. [Create](#) or load a layer with points.
2. Select **Terrain › DEM › Build DEM › From points....** The **Parameters of DEM building from points** parameters window opens.

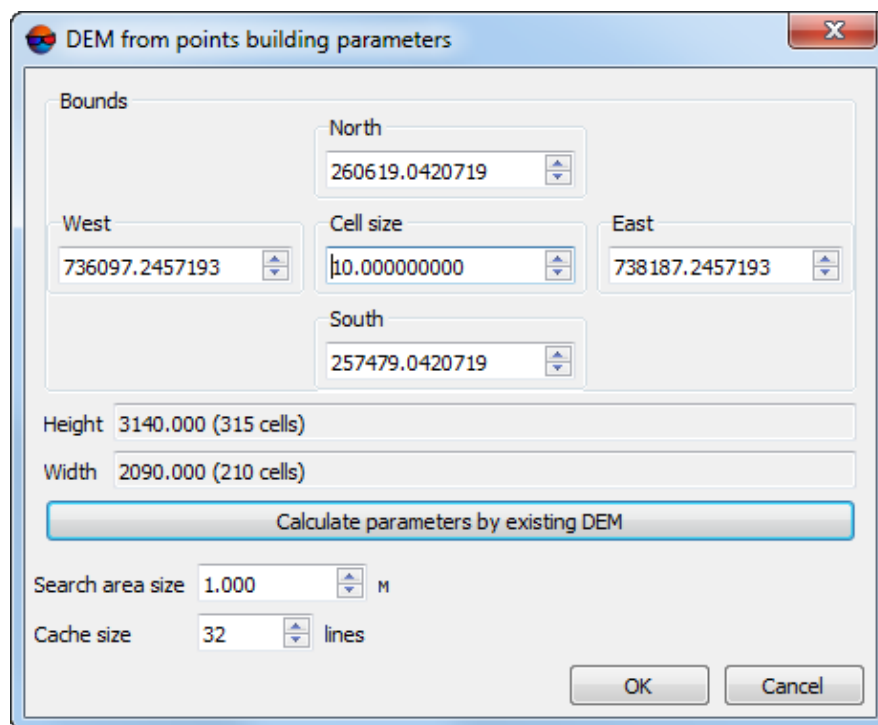


Fig. 117. Parameters of DEM creation from points

- 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, as well as number of DEM nodes.



Coordinates of border points of source layer are default values.

- Specify **Cell size** in meters to change size of output DEM cell.



DEM cell size should be commensurate with the average distance between points. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

- [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.

- Specify **Search area size** in meters to change 'sliding window' size (in meters on terrain).



'Sliding window' method – means moving of some frame over set of points from node to node of DEM considering elevations and weights of points falling into the 'sliding' frame.

A value of DEM cell (elevation), located in the middle of 'sliding window', is define as the weighted mean value of heights of the points in the 'sliding window'. The weight of every point is inversely proportional to the distance between the point and the center of the 'sliding window'.



Cell size of output DEM should not exceed a size of the 'sliding window', specified in the **Search area size** field.

- Specify **Cache size** parameter in lines of search for points in a specified search area by cell size.



Cache size is used to define a number of lines of search area.

- Click OK. Specify DEM file name, define a folder in active profile resources and click **Save** (see also [Section 7.10](#) and [DEM load](#)). 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.

7.3.4. DEM creation using CSV-file

The system provides possibility to create DEM using data obtained from CSV-file.



It is possible to use points coordinates from CSV-file only if points coordinates are recorded using the template Name X Y Z.

To create DEM by CSV-file perform the following actions:

1. Select **Terrain > DEM > Build DEM > From CSV file data...** The **Parameters of DEM building by CSV file** parameters window opens.

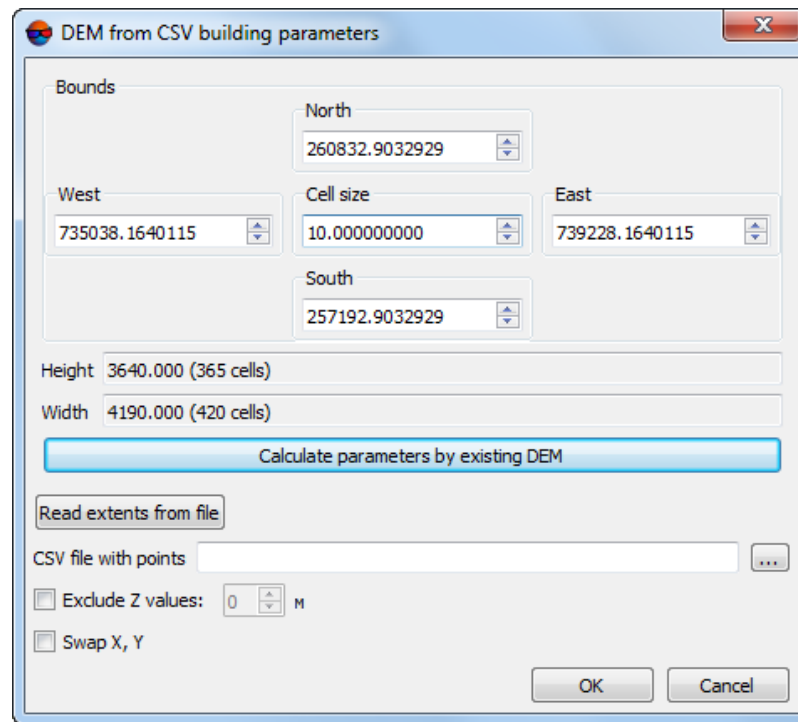


Fig. 118. DEM creation using CSV-file data

2. 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, as well as number of DEM nodes.




Coordinates of border points included to source CSV-file are default values.

3. [optional] To define output DEM border from CSV-file data click the **Read extents from file** button.
4. Specify **Cell size** in meters to change size of output DEM cell.



DEM cell size should be commensurate with the average distance between points. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

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 the  button and select CSV-file to be used for DEM creation.
7. [optional] In order to exclude points with specified elevation from creation operation, set the **Exclude Z values** checkbox on and specify the value in meters.
8. [optional] In order to swap coordinate system (left/right), set the **Swap X, Y** checkbox on. At that the coordinate system do not change.



If the **Swap X Y** checkbox is cleared, the vector objects will be imported in the right coordinate system. Otherwise, in the left one.

9. Click OK. Specify DEM file name, define a folder in active profile resources and click **Save** (see also [Section 7.10](#) and [DEM load](#)). 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.

7.3.5. 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. The method supports a wide range of input data: aerial central projection images, UAS data, pushbroom scanner imagery, VisionMap and ADS data.



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.



When building a dense DEM using the SGM method within the projects of the Satellite scanner imagery and ADS types (see the “Project types” in the “[Creating project](#)” User Manual), it is necessary to create areas without background (see the “Creating areas without background (useful areas)” in the “[Creating project](#)” User Manual)

When processing Satellite scanner imagery or ADS projects, the program checks the presence of areas without background automatically, then, if they are absent, builds them automatically with default parameters or uses the areas without a background previously created by the user, if available.

The areas without background created in such a way can then be edited by the user for further use, e.g. when rebuilding a DEM based on updated data (if appropriate). However, in most cases, the preliminary creation of areas without background and/or their manual correction are not required.

When creating a DEM using the SGM method for projects on the base of central projection aerial imagery (see the “Project types” in the “[Creating project](#)” User Manual), automatic creation of areas without background is not performed. However, if areas without background, for some reason, will be manually created by the user beforehand, they will be taken into account when building a DEM.

This may be necessary in cases when aerial images have areas containing a background, and this, for example, cannot be corrected through the **Offset from image boundaries** customization (see below).

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)...** 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.

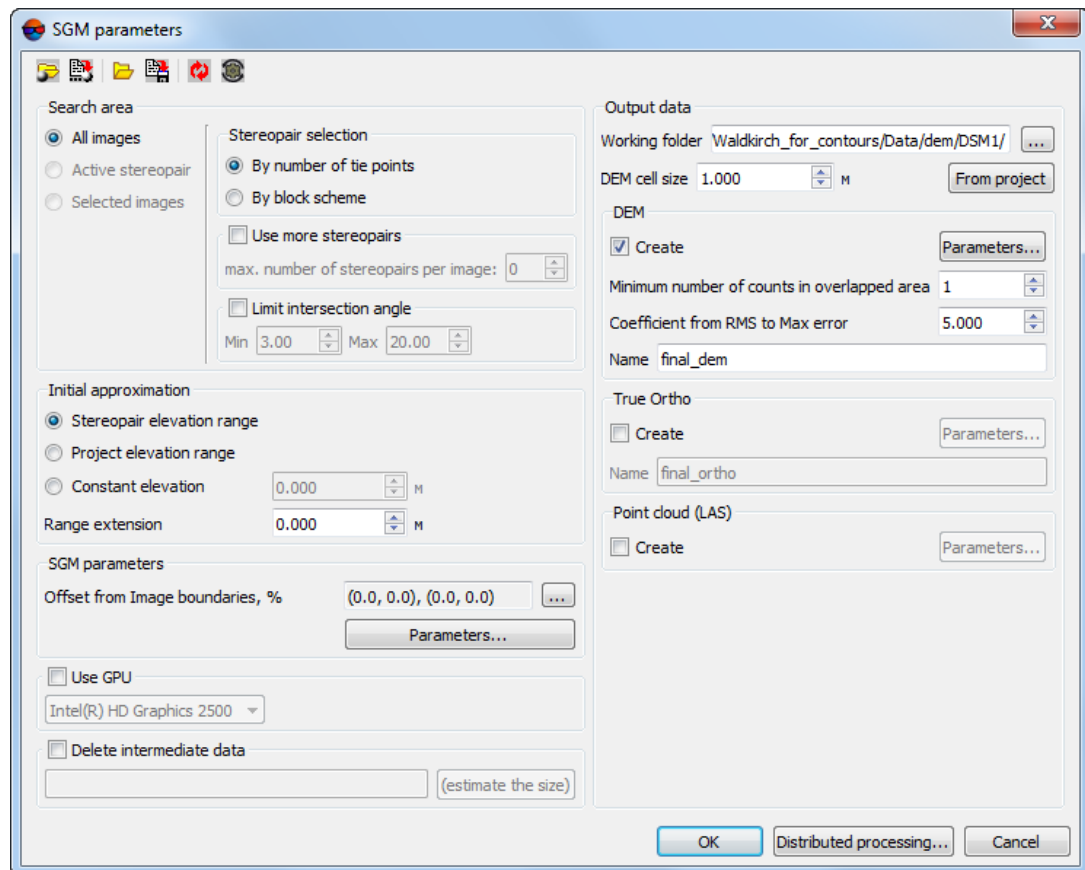








Fig. 119. SGM parameters

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 ori-

entation 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).

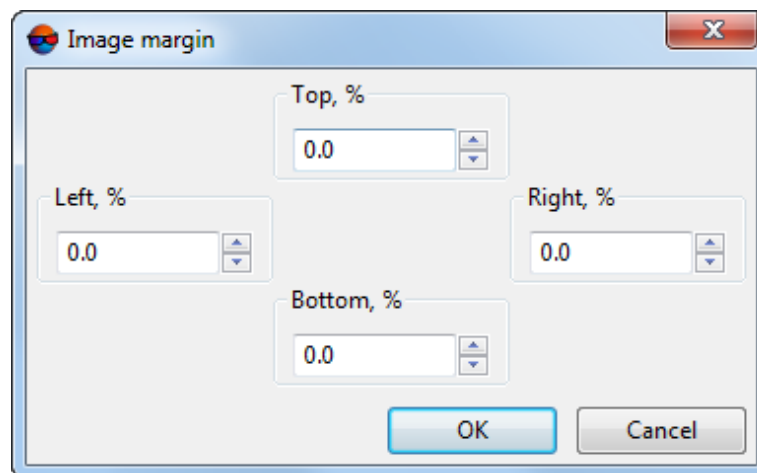


Fig. 120. The Image margin window



Fig. 121. Image margin

8. For more detailed settings of SGM cost calculation click **Additional....** The **SGM parameters** window opens.

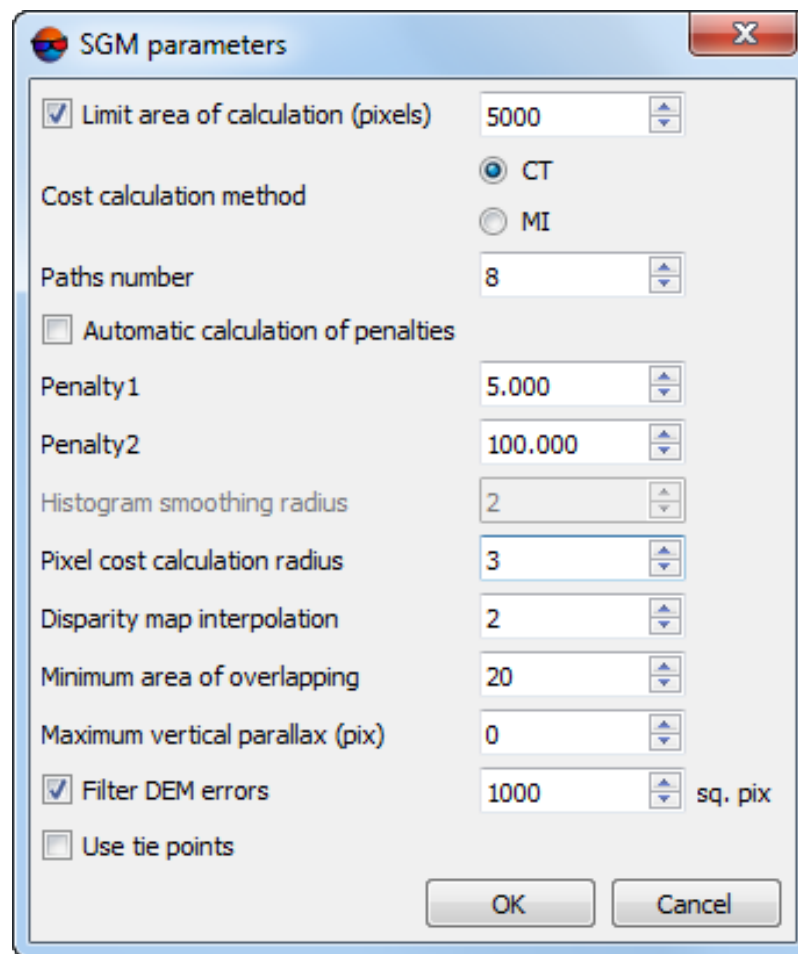


Fig. 122. SGM parameters

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.



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.

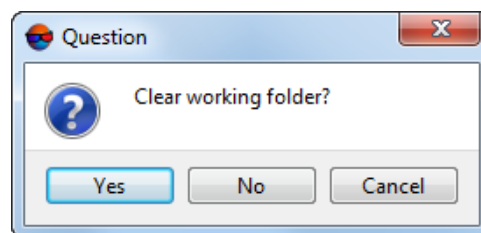


Fig. 123. Question

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 in 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. 124. 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 filename.

14. [optional] To **create** the **True Ortho**, limited by DEM boundaries, set the appropriate checkbox and specify the output **true ortho filename**. 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:

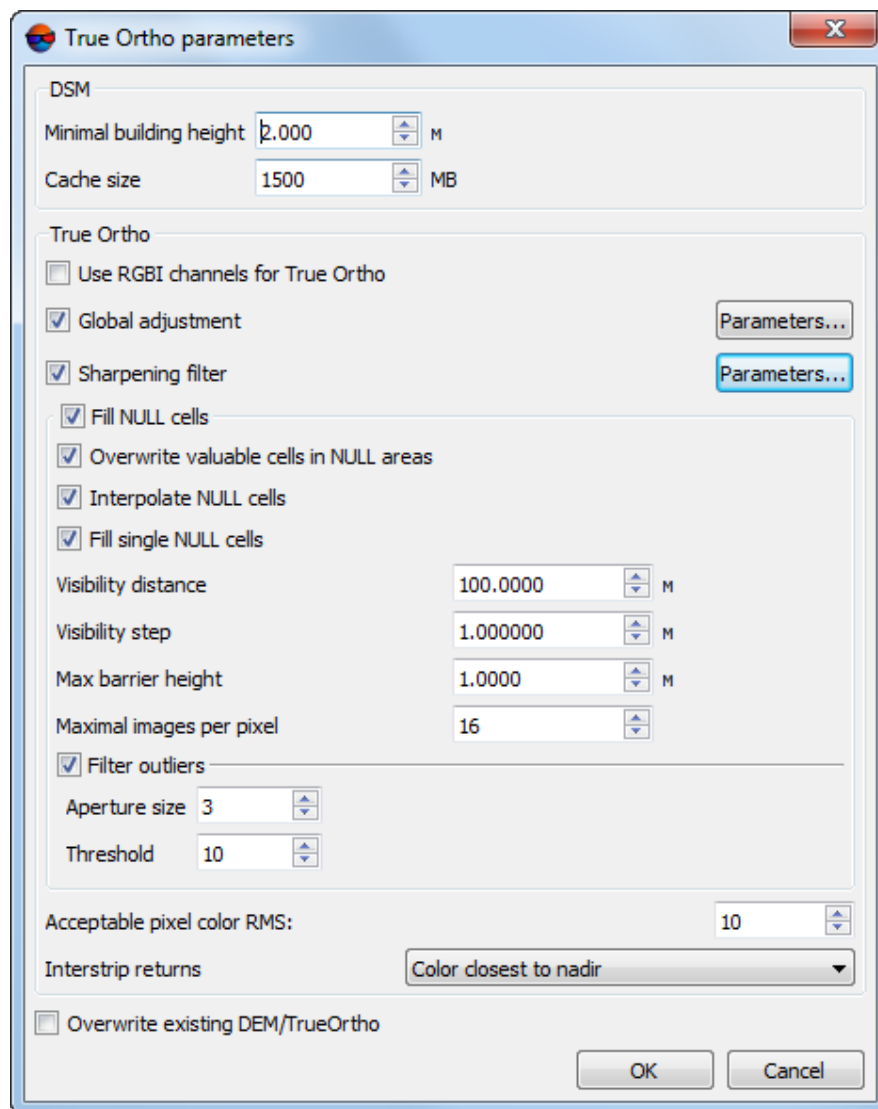


Fig. 125. The Fill NULL cells window

- 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:

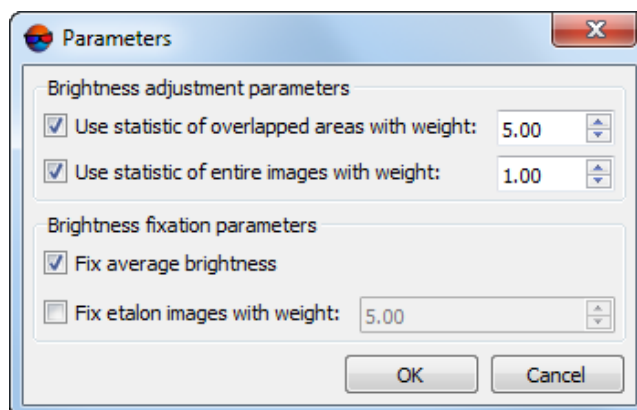


Fig. 126. Brightness adjustment parameters

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.

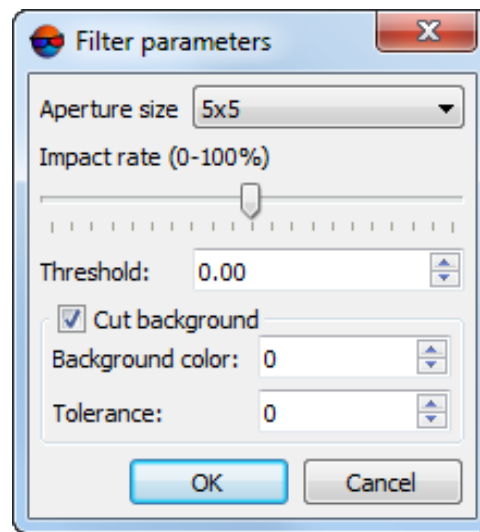


Fig. 127. Filter parameters

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 determined 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.

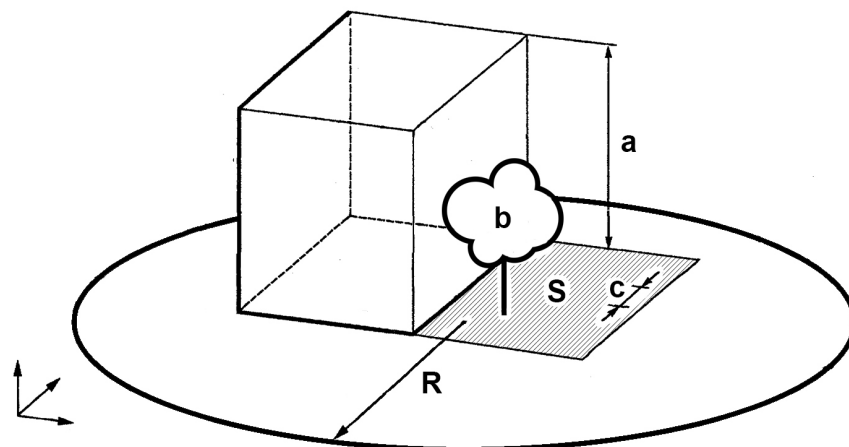


Fig. 128. An example of unfilled area on true ortho, where **a** is the building height (see **Maximal barrier height** parameter), **b** is the “isolated DEM area” among DEM NULL cells (see **Overwrite valuable cells in NULL areas** checkbox), **c** is the **Visibility step**, **R** is the **Visibility 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.

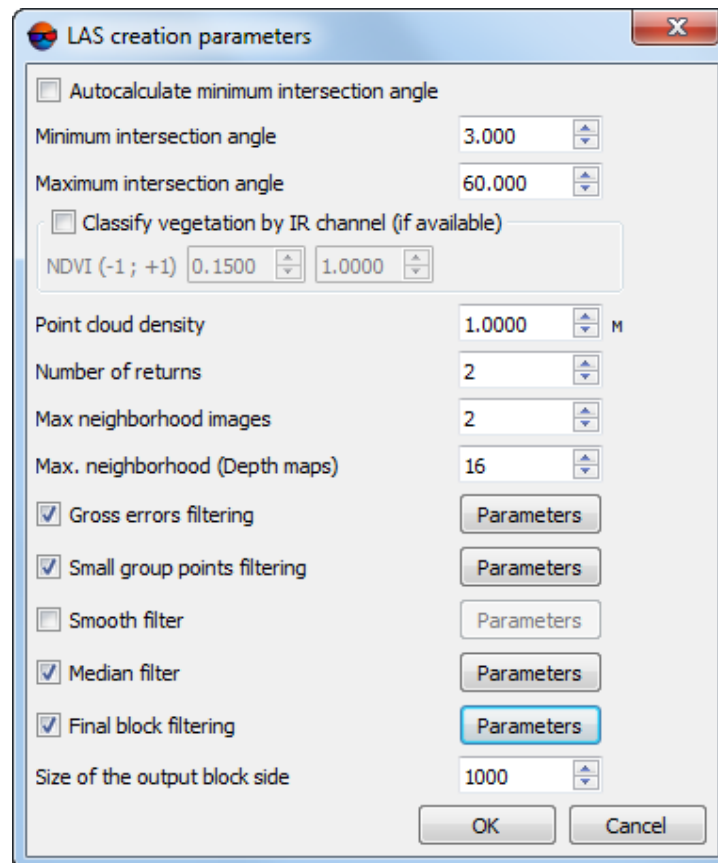


Fig. 129. The Create LAS parameters window

- Set the **Maximum intersection angle** and **minimum 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 an 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**:

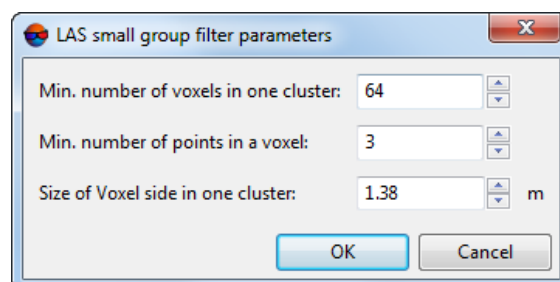


Fig. 130. The “LAS small group filter parameters”

- **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.

7.3.6. DEM creation in batch mode

The system provides possibility to create DEM in batch mode. Batch mode allows to create multiple files with TIN and DEM simultaneously and to save them to specified folder.

For DEM batch creation the source vector resources are divided into sheets using specified parameters. For each vectors sheet the system creates TIN, used for DEMs creation, and then they are merged to generate a single output DEM.

Perform the following actions to execute batch processing:

1. Select **Terrain › DEM › Build DEM › Batch DTM creation**. The **Create batch DTM – Step 1 of 2: Source data** window opens:

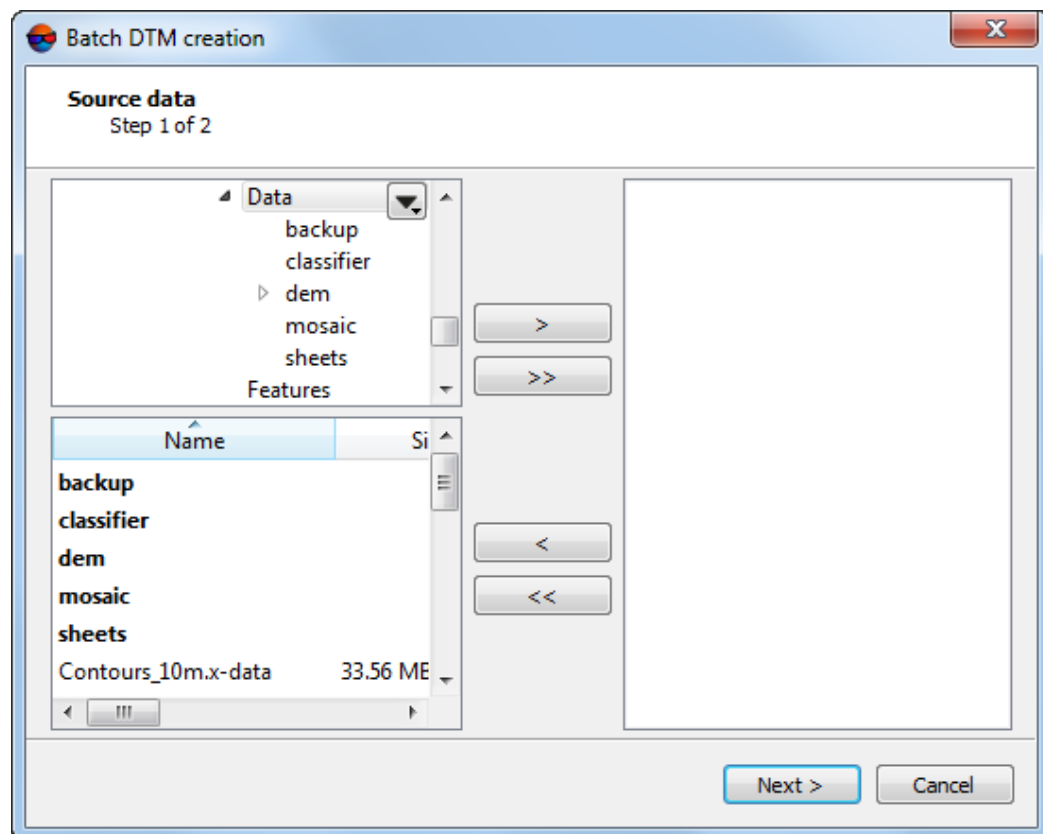












Fig. 131. Creating a list of the initial vector resources for batch DTM creation

2. Choose a folder containing vector objects in resources tree.

 The  button allows to display all available resources in nested files. The  button allows to refresh part of the window with resources.

 The  button allows to show a list with 10 recently selected resources.

3. Select vector file in the list and click the  button to add a layer.

 The  and  buttons allow to add to the list or remove from it all added files with vector objects, the  button allows to remove selected file from the list

4. Repeat the 2 and 3 steps to add next files with vector objects.
5. Click the **Next** button. The **Create batch DTM – Step 2 of 2: Options**.

Batch DTM creation
Step 2 of 2

Options

Actions

1. Split input data into sheets
 - max points per sheet: 1.000 min. points
 - sheets overlap: 50.00 %
2. ☒ Build TIN by created sheets
3. ☒ Build DEM by created TIN
- ☐ Distributed processing

Working folder

All existing resources in the working folder will be deleted before proceeding!

Filtering

☐ Z-range filter

Zmin: 470.000 M Zmax: 770.000 M

DEM parameters

Extents

North: 262063.50 M
West: 735168.79 M East: 739793.79 M
South: 257419.42 M
Height: 4644.080 M
Width: 4625.000 M

Cell size

DEM cell size: 50.000 M From project

Number of cells:

Height: 92
Width: 92

Estimated DEM size: 43.97 KB

Calculate parameters by existing DEM

< Back OK Cancel

Fig. 132. Parameters of batch DTM creation

6. In the **Processing** section define tasks to be performed during DEM creation:

1. **Split input data into sheets** – specify **max.point quantity per sheet** in millions of points, and specify a size of **sheets overlap** in percent.



Specified parameters are considered by the system to define a number of sheets for splitting, so in specified target folder the system creates vector files with *.x-data extension and *sheet_sheet №* names.

2. **Build TIN by created sheets** – (is set on by default) is used to create TIN by vector objects layers, divided into sheets. During execution of this task in the specified target folder the system creates files with *.x-tin extensions and *sheet_sheet №* names, that correspond to vector files.
3. **Build DEM by created TIN** – (is set on by default) is used to create a single DEM by the whole TIN, generated using obtained sheets.



The task is performed in two steps: DEM is created for each sheet by TIN, after that all created DEMs are merged into single one. Thus in the output folder a file with *.x-dem extension is created.

7. [optional] Set the **Distributed processing** checkbox on to use distributed processing feature for batch DEM generation.

8. In the **Working folder** section define **empty** folder in active profile resources, where to place output data.



Before operation is performed all data from selected folder is removed.

9. [optional] In the **Filtering** section set the **Z-range filter** checkbox on to apply filtering to vectors and/or TIN. During filtering the system removes points and vertices of polylines/polygons (including TIN vertices), which Z-coordinate is out of range, specified in the **Zmin** and **Zmax** fields in meters.
10. In the **DEM parameters** section specify DEM boundaries in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters.
11. Specify **Cell size** in meters to change size of output DEM cell. In the **Height** and **Width** fields of the **Cell size** section the system displays nodes number calculated with specified DEM cell size. There is also displayed the **Appr DEM size** in megabytes.



DEM cell size should be commensurate with the average distance between points. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

12. [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.
13. Click the **Ok** button. The system displays a message about removing of all data in specified output folder. After that the system creates new DEM layer in the *Manager*.

When the **Distributed processing** checkbox is set on – change settings and run the distributed processing server/client, before starting the operation (see the “Distributed processing” chapter in the “[General information about system](#)” User Manual).

Click **Ok**. 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**.

7.3.7. Building DEM from an external source

The system provides possibility to build a DEM from an external source (SRTM data) within boundaries of area specified by a polygon.

SRTM (Shuttle Radar Topographic Mission) – data obtained as a result of the international mission to build a digital elevation model (DEM) of the Earth's surface. SRTM data is a DEM, covering the territory of the land from 60°N to 54°S and some parts of the sea.

There are some versions (releases) of SRTM data, differing in the degree of additional processing.

The SRTM data are distributed in two versions:

- **SRTM1** – a matrix with a cell size of 1x1 angular second (i.e. with a spatial resolution of 30 meters) – are available on the US territory only;
- **SRTM3** – a matrix with a cell size of 3x3 angular second (i.e. with a spatial resolution of 90 meters) – publicly available data for the rest of the Earth's surface covered by the mission.



At the moment, the system provides the ability to build a DEM using SRTM data of v4.1 with the cell size 3x3 angular seconds (SRTM3 v4.1), presented in *.zip archive.

The SRTM data are distributed in *tiles* – fragments of SRTM DEM, with size 1x1 or 5x5 degrees, presented in *.zip archives. For the identification of tiles, mesh-grids are used in the Shape format with step 1 or 5 degrees, respectively.



Prior to DEM creation from an external source within the selected polygon, you should do the following:

- set global working coordinate system (see the “Coordinate systems” and “Project properties” chapters in the “[Project creation](#)” User Guide). Otherwise, during DEM creation the system displays a corresponding error message;
- import to the *PHOTOMOD* system a Shape file of the corresponding mesh-grid (see the “Import from Shape” chapter of the “[Vectorization](#)” User Guide) and save it to the format used in *PHOTOMOD* system.

To create DEM an external source within the selected polygon, perform the following:

1. [optional] Load or create a new polygon, which is used as the created DEM border (see below). Select the polygon;
2. Choose **Terrain › DEM › Build DEM › Create DEM from external source bounded by current polygon....** The **DEM creation parameters** window opens:

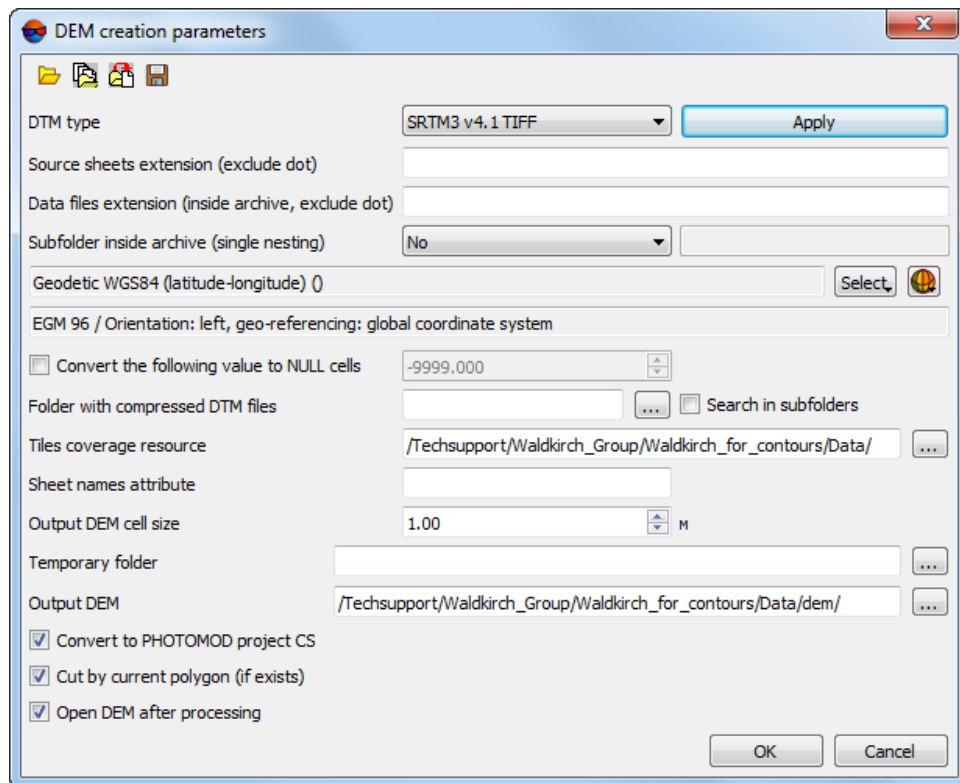


Fig. 133. Parameters of DEM creation from an external source

The toolbar in the upper part of the window is intended to save/load **DEM creation parameters** and input/output data paths.

It contain the following buttons:

- – allows to load DEM creation parameters and data paths from *.x-ini file located in a folder of file system;
- – allows to load DEM creation parameters from *.x-ini file located in a folder of file system;
- – allows to load data paths from *.x-ini file located in a folder of file system;
- – allows to save DEM creation parameters and data paths to *.x-ini file located in a folder of file system.

Specify the following settings:

- Select the **DTM type** (a SRTM version, which used for DEM generation);

Click **Apply** to automatically specify the coordinate system of the input data, the **Source sheets extension** (i.e. the archives containing them), and the **Data files**

extension (contained within the archives themselves), according to the **DTM type** selected.

If necessary, input the required data manually.



SRTM data is presented in WGS 84 coordinate system.

- [optional] Specify the **single nesting subfolder inside archive** with *.zip extension, where SRTM data are stored;
- [optional] to **convert the following value to NULL cells**, set the appropriate checkbox and specify DEM cell height;
- Specify the **folder with compressed DTM files**, where SRTM data are stored (presented in *.zip format archives);
 - [optional] set the **search in subfolders** checkbox to perform a search for *.zip archives with SRTM tiles not only by specified path, but also in all its subfolders.
- Specify a path to preliminary imported **tiles coverage resource**, containing mesh-grid (see above), in format used in the *PHOTOMOD system*;
- Input the **attribute with sheet name** – an attribute *name*, containing names of archives with SRTM data. The required attribute name is included to attributes of a vector layer, obtained during import of mesh-grid (see the previous step);
- Set the **output DEM cell size**, in meters;
- Specify a **temporary folder** in the active profile resources, to store temporary files;
- Specify a folder in active profile resources to save **Output DEM**;
- [optional] set the **convert to PHOTOMOD project CS** checkbox, to recalculate an output DEM into coordinate system of the current project;



SRTM data is presented in WGS 84 coordinate system.

- [optional] set the **cut by current polygon** checkbox (see paragraph 1);
- [optional] set the **Open DEM after processing** checkbox, if needed.

3. Click OK.

7.4. DEM filtering

The system allows to filter DEMs. The following filters are provided for editing DEMs:

Table 7. Brief description of the “Filter” menu

Menu items	Function
Buildings and vegetation filter	to Buildings and vegetation filtering in DEM
Slope based filter	to filter objects based on slope angle
DEM classification	to classify the DEM cells . Classification is performed through the joint analysis of a DEM and an ortho-photo that has an IR band
Extract classes from DEM	to save the selected groups of pre-classified DEM cells into a separate resource
Median filter	to use median filtering of DEM cells
Smooth filter...	to use smoothing filter for DEM values
Bilateral filter	the bilateral filter successfully suppresses small noises, preserving the characteristic features of the terrain, and therefore it is recommended to use it before building contours using DEM (after filling in null cells)
Filter by image properties	to filter DEMs depending on image characteristics
Cut DEM by Z-range	to convert DEM cells with specified elevation to null cells

7.4.1. Buildings and vegetation filter

The system allows to use filtering of buildings and vegetation of DEM. Such filtering includes calculation of base points with specified coefficient of DEM subsampling. The operation includes deletion, correction or recognizing points which have fallen on buildings, trees, cars, in pits, etc. during [automatic points calculation](#), as well as filtering of accident spikes. After the filtering output DTM contains just terrain relief model.

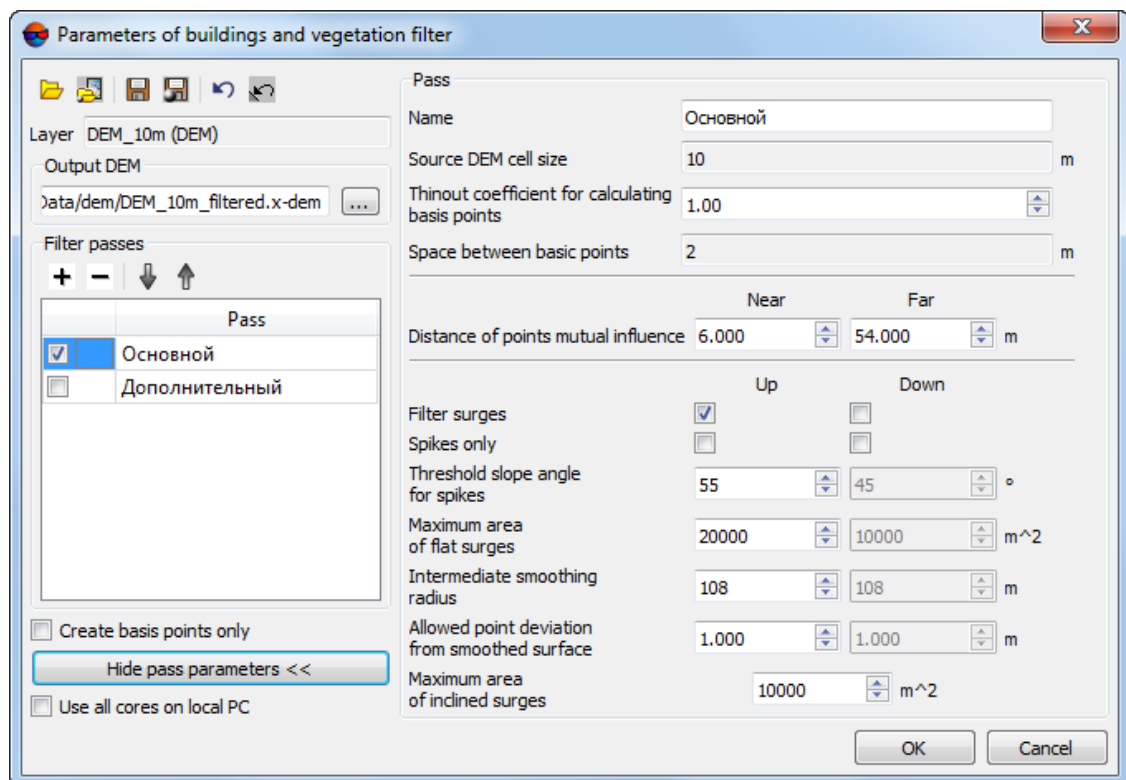


Fig. 134. Buildings and vegetation filtering from DEM

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.

Button	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
	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 buildings and vegetation on DEM perform the following actions:

1. Make active a DEM layer.
2. Select **Terrain > DEM > Filter > Buildings and vegetation filter**. The **Buildings and vegetation filter** window opens.

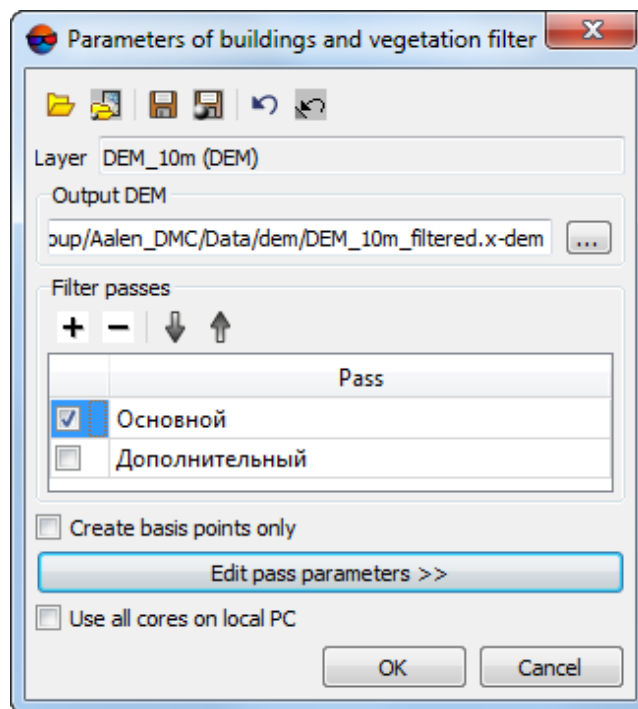


Fig. 135. Buildings and vegetation filtering from DEM

3. In the **Output DEM** section click the button and specify name and target folder of output DTM in active profile resources.
4. [optional] Specify a number of filter passes:
 - by default it is used two passes with standard parameters – **Basic** and **Additional**. Set off checkboxes to change number 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.
5. Select a name of pass and click the **Edit pass parameters** button to display and configure the following parameters of filter pass:

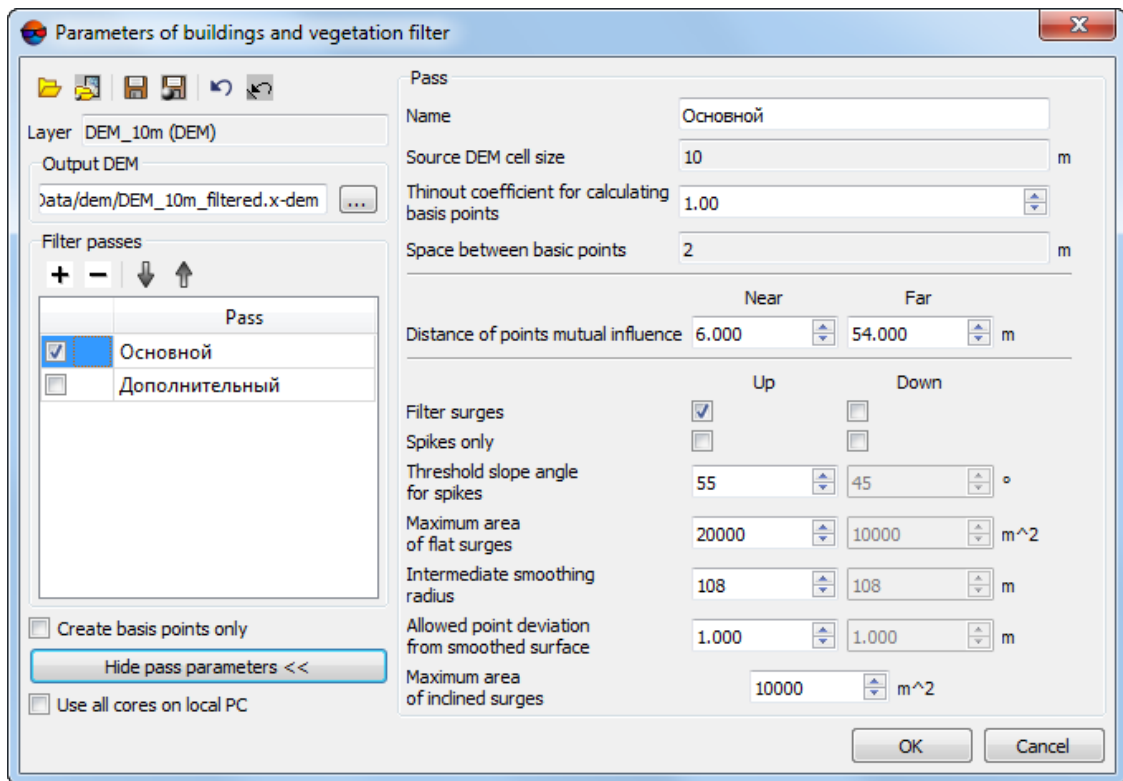


Fig. 136. Configuration of buildings and vegetation filter

- [optional] to change a filter pass name, input new name to the **Name** field;
- the **Source DEM cell size** field shows DEM cell size in meters before filter applying;
- specify **Thinout coefficient for calculating basic points** by DEM. From the specified value and the **Source DEM cell size** parameter depends the **Space between basic points** parameter in meters;
- 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.

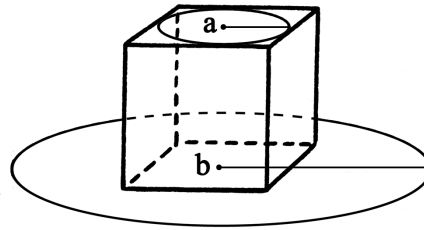


Fig. 137. Near distance of points mutual influence (a) and far distance of points mutual influence (b)

- 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 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).

6. [optional] To use all cores of used computer, 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.
7. 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 points being filtered.

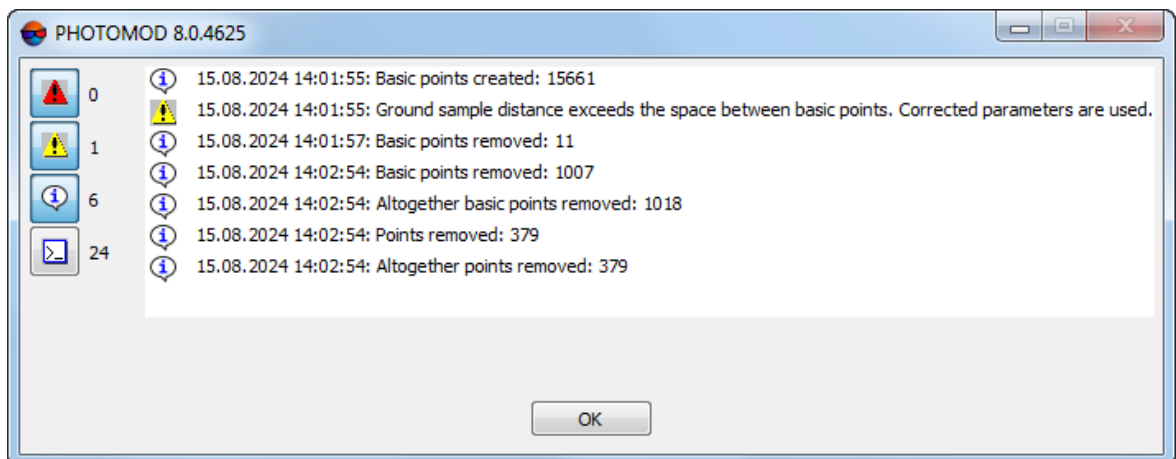


Fig. 138. Filtering results

7.4.2. 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....** The **Slope based filter** window opens;

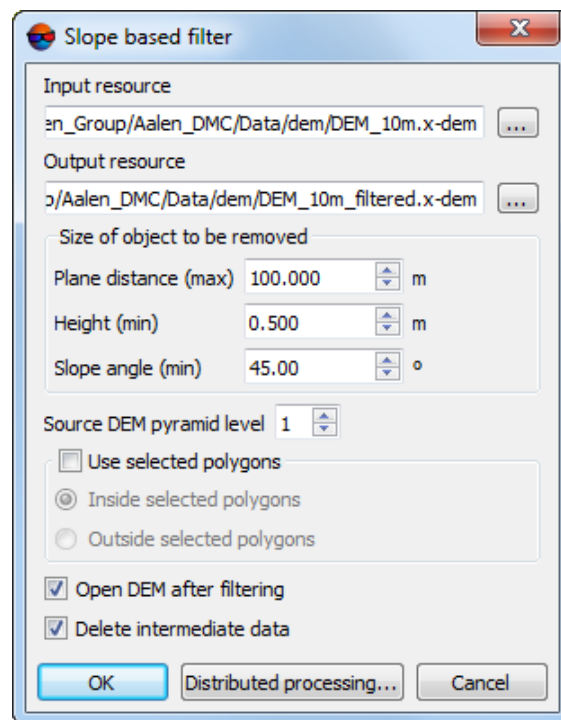




Fig. 139. Parameters of slope based filter

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.

7.4.3. Median filter

The system allows to use median filtering of DEM cells.

The *median filtering* contains the following sequence of actions:

1. Sequential scanning of DEM by mask with specified size. Size of scanning mask is defined by the **Aperture** parameter (in DEM cells).
2. The system replaces current cell elevation values by median values, if a difference between them is more than specified **Threshold** in meters.

To apply median filtering perform the following actions:

1. Make active a DEM layer.
2. Select **Terrain > DEM > Filter > Median filter**. The **Median filter parameters** window opens.

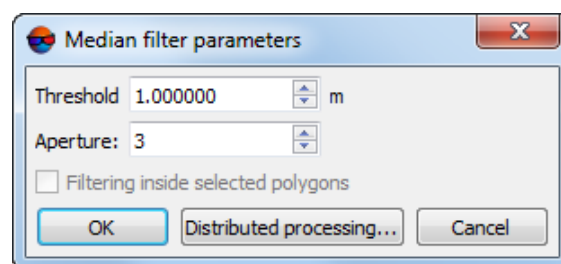


Fig. 140. Median filter parameters

3. In the **Threshold** field specify a value of threshold in meters, which is used to compare difference value between current and median elevation.
4. In the **Aperture** field specify a size of scanning mask.
5. [optional] Set the **Filtering inside selected polygons** checkbox if needed.
6. Click OK. After that current values of DEM cells are replaced by median values, if difference value between current and median elevation does not exceed specified **Threshold**.

To use distributed processing for DEM median 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.

7.4.4. Smooth filter

The system allows to use smooth filtering of DEM cells values.

To execute smooth filtering perform the following actions:

1. Make active a DEM layer.
2. Select **Terrain › DEM › Filter › Smooth filter**. The **Smooth filter parameters** window opens.

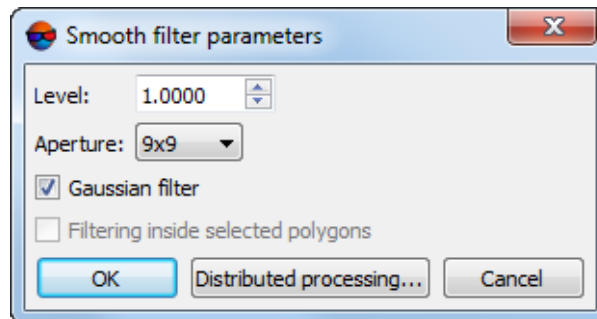


Fig. 141. Smooth filter parameters

3. In the **Level** field specify filtering coefficient value from 0 to 1.
4. In the **Aperture** field specify a size of scanning mask.
5. [optional] To apply Gauss smoothing algorithm the **Gaussian filter** checkbox is set on by default.
6. [optional] Set the **Filtering inside selected polygons** checkbox if needed.
7. Click OK. After that current values of DEM cells elevation are replaced by filtered values.

To use distributed processing for DEM smooth 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.

7.4.5. Bilateral filter

The system provides for smoothing DEM filtering using a bilateral filter. The bilateral filter successfully suppresses small noises, preserving the characteristic features of the terrain, and therefore it is recommended to use it before [building contours](#) using DEM (after [filling in null cells](#)).

1. Make active a DEM layer;

2. Select **Terrain > DEM > Filter > Bilateral DEM filter**. The **Bilateral DEM filter** window opens:

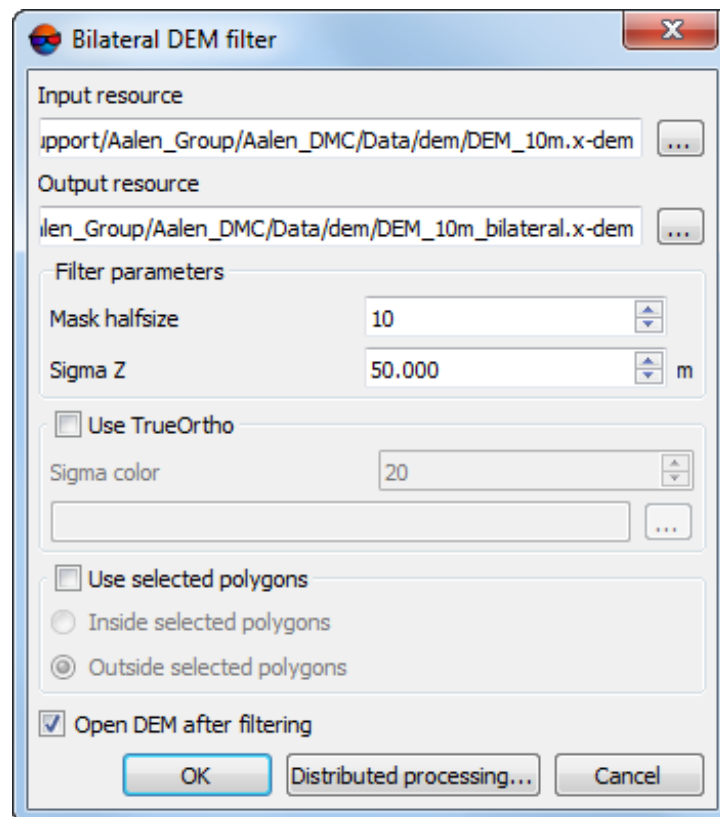


Fig. 142. Parameters of bilateral filter

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 the **Mask halfsize** field specify a size of smoothing mask;
6. Set the **Sigma Z** value that determines the influence of elevations in a mask on the smoothing process;
7. [optional] set the appropriate checkbox to **use true ortho** and click the button to select orthophotomap in resources of the active profile;
 - Set the **Sigma color** value that determines the influence of true ortho pixel colors on the DEM filtering process.
8. [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.

9. [optional] Set **Open Dem after filtering** checkbox to open output DEM in a new layer immediately after filtration completion.
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.

7.4.6. Filter by image properties

The system allows to filter DEMs depending on image characteristics. At the preliminary step (so-called “Learning”) an operator manually collects specific areas of images (in example images – areas covered by forestry) as samples for filtering.

To filter DEM, perform the following:

1. Make active a DEM layer;
2. Select an image by characteristics of which filtration will be done in 2D-window or in **Block editor** window. Select **Window › New 2D-window (stereopair) (Ctrl+Alt+W)** or click button  in the main toolbar. 2D-window (stereopair) is opened;



Filter by image properties function works correctly in a stereomode only.

3. Select **Terrain › DEM › Filter › Filter by image properties**. The **Filter by image properties** window opens.

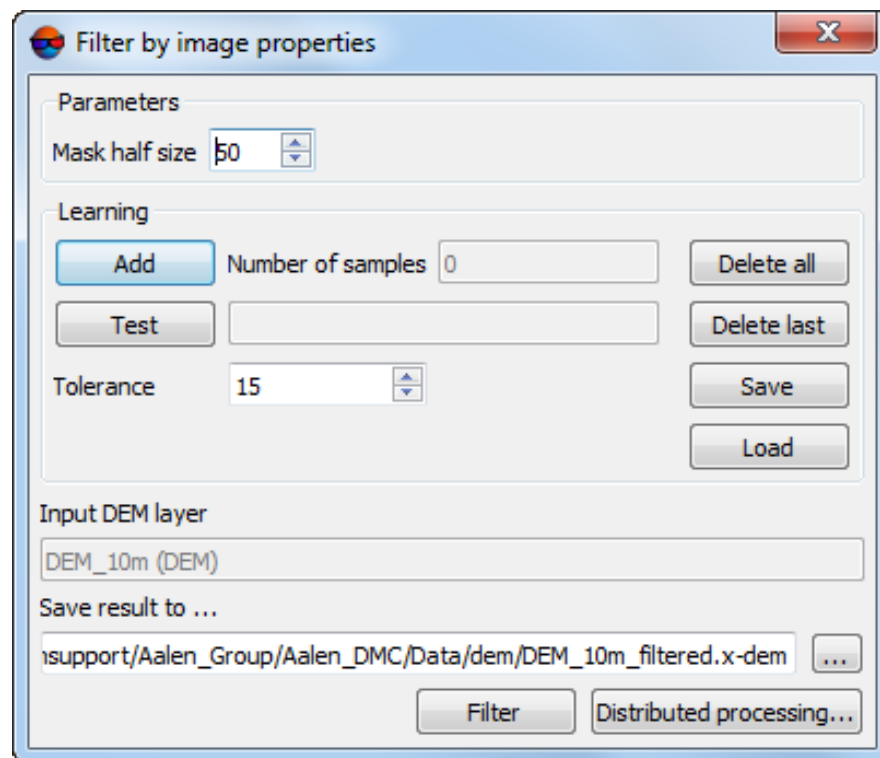


Fig. 143. Parameters of filtration by image properties

4. In **Parameters** section set **Mask half size** – a size of square mask-window scanning DEM;
5. Move marker into specific image area and click **Add** button;



Delete last button is designed to delete the latest collected sample **Clear all** button is designed to delete all the collected samples.

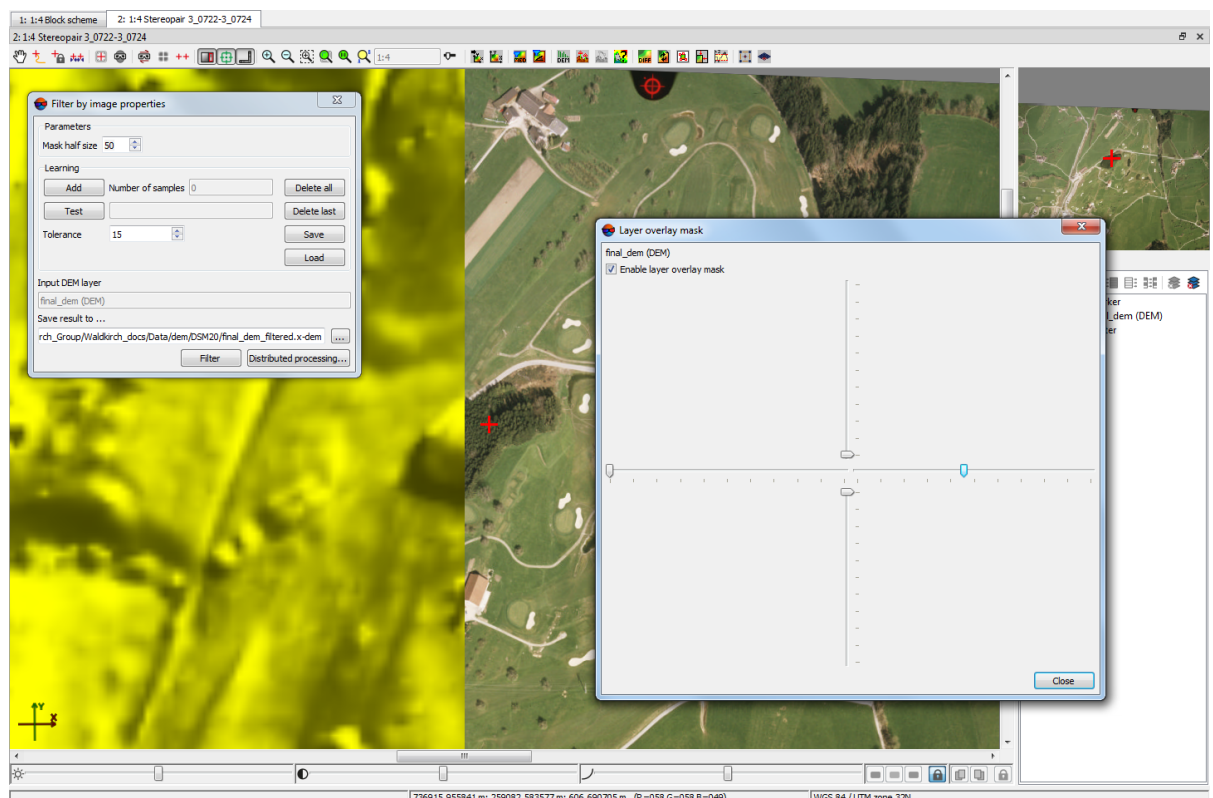



Fig. 144. Selection of the samples

6. Repeat actions described in item 2 several times while changing a marker position;



To check learning results move marker into image area similar or dissimilar to samples areas and click **Test** button. “-” or “+” results in a status box indicate matching or mismatching of tested area characteristics with characteristics of samples areas.

7. In **Save result to...** section click button  and define the name and target folder to save output DEM in resources of the active profile;
8. Click **Filter** button. After finishing filtration process the output DEM is saved in resources of the active profile;

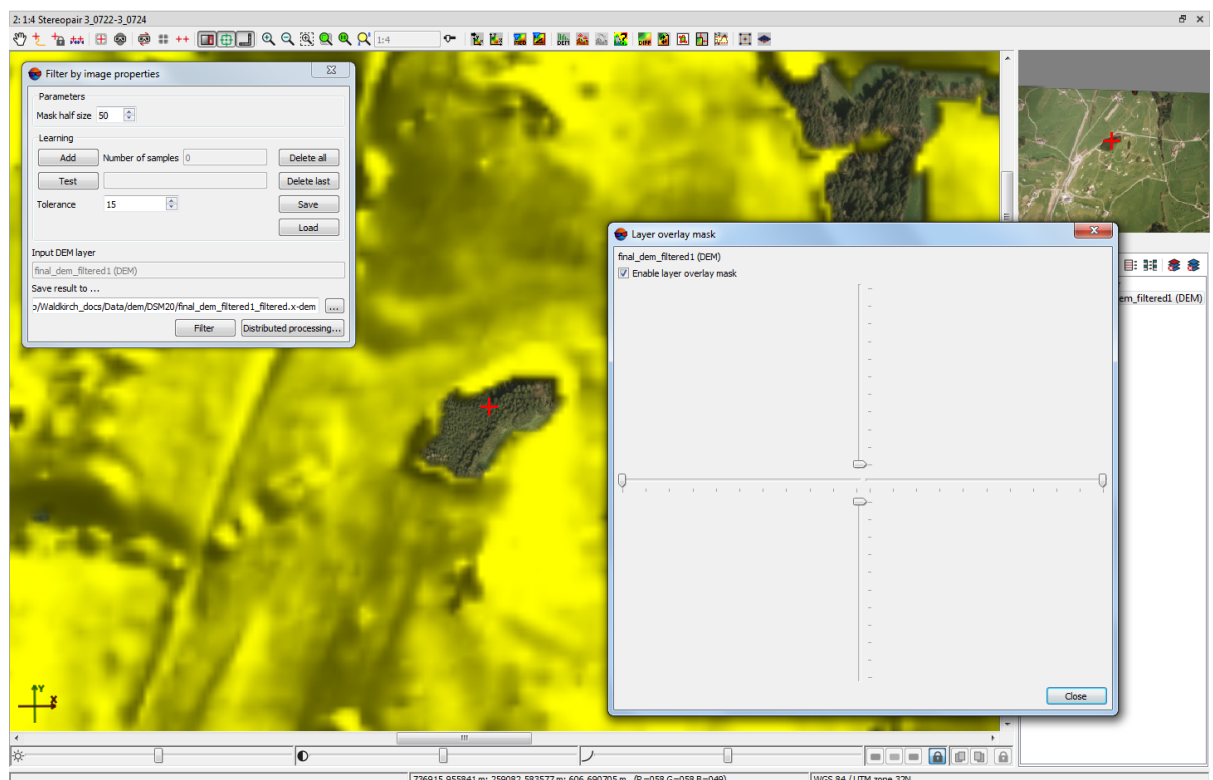


Fig. 145. DEM after filtration by image properties

To use Distributed processing mode while filtering DEM by image properties, 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 **Distributed processing** button. Distributed processing parameters window is opened;

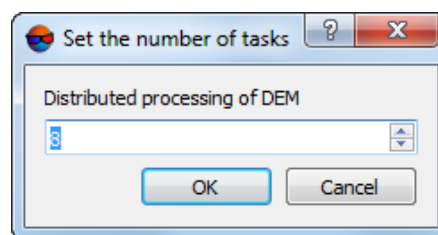


Fig. 146. Parameters of distributed processing

3. Set the **number of tasks**. It is recommended to set **Tasks number** as 1 task per 1 PC.



4. Click OK. Distributed processing tasks are created and the report on the number of tasks created appears.

7.4.7. Selective filtering of objects

The system provides for selective filtering of DEM areas that describe houses, trees, cars, and other similar objects. After applying the filter, the processed section of the DEM describes only the terrain.


The tools for selective filtering of objects are located on the **DEM editing** toolbar. The system also provides for pre-selection of boundaries of the object to be removed from the DEM's surface.

To preselect a DEM segment which describes an object that is supposedly not a part of the terrain, without deleting it, perform the following:

1. Make the DEM layer active;
2. Choose **Window › Toolbars › DEM edit**. The **DEM editing** toolbar opens;
3. Click  on the **DEM editing** toolbar to preselect objects. The shape of the cursor () indicates the enabled mode of preselection of objects;



Click **left mouse button** once to refresh the cursor.

4. [optional] Click  on the **DEM editing** toolbar. The section to set object search parameters opens:

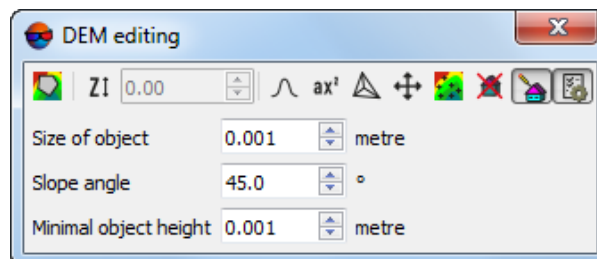


Fig. 147. The “DEM editing” toolbar

Specify:

- The estimate **size of object** to be deleted, in projects units;
- **Slope angle**, in degrees.







The **slope angle** is one of the parameters by which the filtering start threshold is calculated. For example, for a building with a sheer wall, the slope angle will be ~ 90 degrees


(note, that due to the peculiarities of constructing DEMs, this angle will never be exactly 90 degrees).

The **slope angle** is selected individually for each specific case according to the characteristic buildings and objects that need to be identified. Large values of the slope angle (about 80 degrees) can lead to the fact that buildings of a specific shape (for example, hangars) may not be removed during the filtering. Reducing the slope angle (for example, to about 45 degrees) can lead to the fact that, along with the above buildings, some terrain details can also be removed (for example, hills of similar shape and size).

- **Minimal object height** over the surface, in project units.

5. Make sure, that the **Subtract from selection**  or **Invert selection**  modes are not enabled in appropriate toolbar (**Window > Toolbars > Tools**). Otherwise – enable the **Normal**  or **Add to selection**  mode;
6. Place the cursor on the desired object on the DEM surface and start recognition double-clicking the **left mouse button**. The progress window opens. Wait for the recognition operation to complete and the progress window to close.


If a section of the DEM describing an object that is presumably not a part of the terrain is found in the vicinity of the marker, then, after the operation is performed, the boundaries of this DEM section will be visually highlighted in the 2D window.

To separate the specified area from the rest of the DEM surface, the border and green fill are used. If the object point filtering mode  was enabled, then the filter would be applied to the object.

If a suitable object is not found in the vicinity of the marker, the No objects for filtering message appears to the right of the displayed current marker coordinates in the **Status** bar in the lower part of the 2D window.

7. [optional] To add one more object to the selected one, repeat the steps described in the above subsection. To deselect the recent object, press **Esc**.





The **Esc** key allows to deselect only one recent object. To deselect all objects, press this key the appropriate number of times. The key works only when the object selection mode is on .



Object selection and filtering modes are not interrelated. If several objects were selected, and then the filtering mode was turned on, only the object on which the marker was hovered will be deleted by double-clicking the left mouse button, regardless of which objects were previously selected.


In order to subject a separate section of the DEM to selective filtering, perform the following:

1. Make the DEM layer active;

2. Choose **Window › Toolbars › DEM edit**. The **DEM editing** toolbar opens;
3. Click  on the **DEM editing** toolbar to enable the object selective filtering mode. The shape of the marker () indicates the enabled object point filtering mode;



Click **left mouse button** once to refresh the cursor.

4. [optional] Click  on the **DEM editing** toolbar. The section to set object search parameters opens:

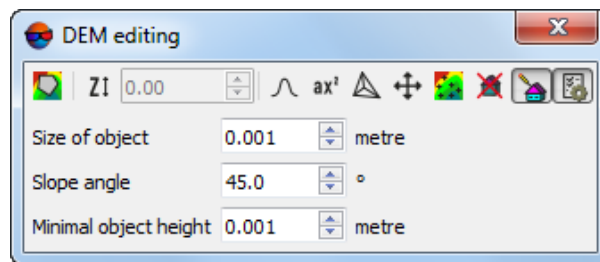


Fig. 148. The “DEM editing” toolbar

Specify:

- The estimate **size of object** to be deleted, in projects units;
- **Slope angle**, in degrees.



The **slope angle** is one of the parameters by which the filtering start threshold is calculated. For example, for a building with a sheer wall, the slope angle will be ~ 90 degrees (note, that due to the peculiarities of constructing DEMs, this angle will never be exactly 90 degrees).




The **slope angle** is selected individually for each specific case according to the characteristic buildings and objects that need to be identified. Large values of the slope angle (about 80 degrees) can lead to the fact that buildings of a specific shape (for example, hangars) may not be removed during the filtering. Reducing the slope angle (for example, to about 45 degrees) can lead to the fact that, along with the above buildings, some terrain details can also be removed (for example, hills of similar shape and size).

- **Minimal object height** over the surface, in project units.
5. Place the cursor on the desired object on the DEM surface and start recognition double-clicking the **left mouse button**. The progress window opens. Wait for the recognition operation to complete and the progress window to close.

If a section of the DEM describing an object that is presumably not a part of the terrain is found in the vicinity of the marker, then after the operation is performed,


this object will be deleted, and the processed fragment of the DEM will describe only the terrain.

If a suitable object is not found in the vicinity of the marker, the No objects for filtering message appears to the right of the displayed current marker coordinates in the **Status** bar in the lower part of the 2D window.

6. The system provides for canceling some of the operations placed in the **DEM editing** toolbar. Standard functions of the action log are used for this (, , and  buttons of the main toolbar).

The undo log is described in detail in “Editing of active layer” of “[General information](#)” User Manual and its settings are given in “Settings of undo parameters” of “[General system’s parameters](#)” User Manual.

A separate list of actions that can be cancelled is maintained for each layer. It is impossible to undo if too many operations are run automatically. In this case, the appropriate warning appears, and the list of executed operations is not maintained.

If the object filtering operation is canceled, the restored object will be highlighted against the background of the rest of the DEM surface using a border and green fill (see above). To deselect the object, enable the object selection mode () and press **Esc**.

7.5. DEM classification

The system provides for the classification of DEM cells. DEM classification is performed according to the preset classes provided in the *PHOTOMOD* system:

- **Low vegetation;**
- **Medium vegetation;**
- **High vegetation;**
- **Water;**
- **Ground;**
- **Buildings / Bridges;**
- **Small buildings / Cars.**

To see the key of colors of the preset classes in *PHOTOMOD* system, choose **Window › DEM classifier table**. The **DEM classes** window opens:

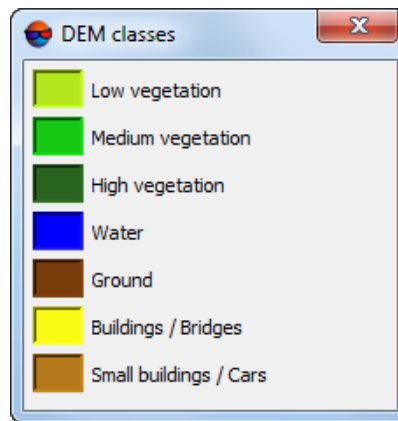


Fig. 149. Preset DEM classes

Classification is performed through the joint analysis of a DEM and an orthophoto that has an IR band and spatially coincides with the DEM under study.

The DEM analysis is carried out using an algorithm similar to the [slope based filter](#) and allows to identify such objects as buildings, trees, cars, etc. (without deleting them in this case).

The NDWI-based orthophoto analysis allows to identify areas of terrain representing the waterplane. The NDVI-based orthophoto analysis allows to identify areas of terrain covered by vegetation.



NDVI is the normalized difference vegetation index. *Vegetation index* is a numerical indicator of the quality and quantity of vegetation in the studied area.

NDVI is one of the most widely used vegetation indices. It is the ratio between the difference in the intensities of reflected light in the red (Red) and near infrared (NIR) bands and their sum:

$$NDVI = (NIR - RED) / (NIR + RED)$$



NDWI is the normalized difference water index. *NDWI* is used to monitor changes related to water content in water bodies, using green and NIR wavelengths:

$$NDWI = (Green - NIR) / (Green + NIR)$$



To classify a DEM according to the NDVI index and NDWI index, the input orthoimage must have an IR channel. To [create](#) an orthoimage, set the **Create** checkbox in the **True Ortho** section of the **SGM parameters** window.

To record IR-band data into the orthoimage under creation, click the **Parameters** button in **True Ortho** section. The **True Ortho parameters** window opens. Set **Use RGBI channels for True Ortho** checkbox in **True Ortho parameters** window and click OK. The creation of True Ortho using four bands is available if the images used as initial data contain the fourth channel, for example, infrared (RGBI).



The system also provides for the classification of a point cloud according to NDVI, that is carried out in a similar way (see “Vegetation classification” in the [“LIDAR Data processing”](#) User Manual).

7.5.1. DEM cells classification

To classify DEM cells perform the following:

1. Make active a DEM layer;
2. Select **Terrain** > **DEM** > **Filter** > **DEM classification....** The **DEM classification** window opens;

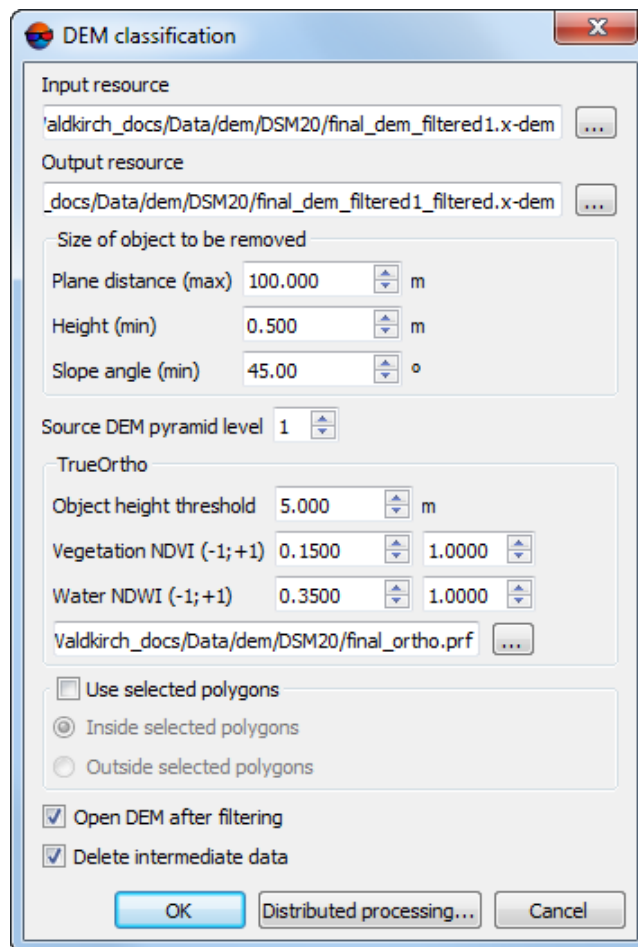




Fig. 150. The DEM classification window


3. In **Input DEM** section click  button and select DEM to classify 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. To perform initial analysis 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 classification. Recommended values of **Intermediate DEM pyramid level** are 1, 2, or 3.



To set zero value is strongly discouraged.

6. Click the  button in **TrueOrtho** section to select an orthoimage spatially coinciding with the DEM under processing, in active profile resources;



To classify a DEM according to the NDVI index and NDWI index, the input orthoimage must have an IR channel.

7. [optional] If required, correct the default **water** class value range in the input fields corresponding to **Water NDWI (-1;+1)**;



Common water range is 0.35 and more. This threshold is used to avoid creating false alarms in urban areas.

8. [optional] If required, correct the default value range in the input fields corresponding to **Vegetation NDVI (-1;+1)**;



A single range is configured for all kinds of vegetation, low, medium, and high.



Common value ranges: from -1 to 0 are for infrastructure objects, as well as for non-wildlife objects (e.g. snow, water, sand, stones, houses, roads, etc.). Common vegetation range is between 0 and 1.

9. [optional] If required, change the default value range in the **object height threshold** input field that defines the boundary between high and medium vegetation;

10. In **Size of object to be classified** section specify the typical anthropogenic object size on the territory to be researched in following fields:

- **Plane distance** (max);
- **Height** (min);
- **Slope angle** (min) in degrees.



A **slope angle** is one of the parameters used to calculate a threshold for classification. 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 classified. 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 classified

as a buildings during the operation. Decreasing the slope angle (e.g. to 45 degrees) may lead to the wrong classification of some accidents of terrain (e.g. elevations having similar shapes and sizes) along with abovementioned buildings.

11. [optional] Set **Use selected polygons** checkbox to classify DEM in the borders of selected polygons only. Select where operation will be performed:

- **Inside selected polygons;**
- **Outside selected polygons.**



To **Use selected polygons**, before opening **DEM classification** 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 classification operation.

12. [optional] Set **Open Dem after classification** checkbox to open output DEM in a new layer immediately after operation completion.
13. [optional] To remove processing data after the calculation is completed set the **Delete intermediate data** checkbox on.
14. Click OK. A process of DEM classification is started. Output DEM is opened in a new layer if the corresponding checkbox was turned on.

To use distributed processing for DEM classification, 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.

7.5.2. Extraction of classes from DEM

The system provides for extracting classes from a [pre-classified](#) DEM. The extraction of classes means the selective saving of selected groups of DEM cells into a separate resource.

For this, perform the following:

1. Make active a DEM layer;

2. Select **Terrain > DEM > Filtering > Extract classes from DEM....** The **Extract classes from DEM** window opens;

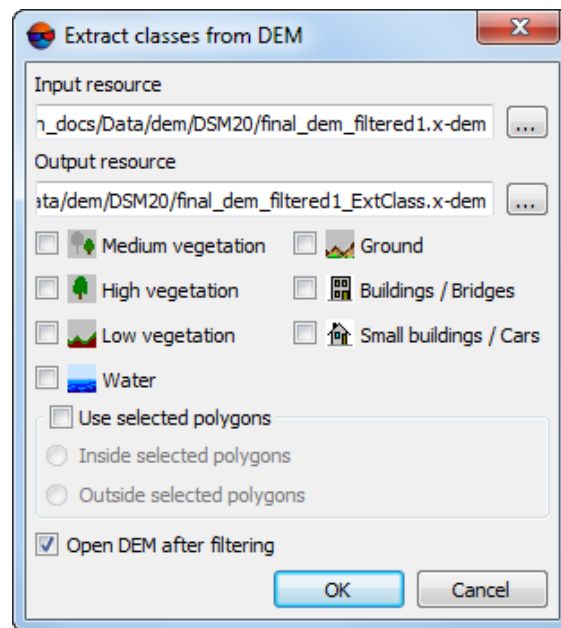




Fig. 151. The Extract classes from DEM window

3. In the **Input resource** section, click  and select a DEM to analyze from the active profile resources;
4. In the **Output resource** section, click  and define a name and folder to store output DEM in the active profile resources;
5. Select classes of DEM cells that are to be saved in a separate file by setting the appropriate checkboxes;
 - **Low vegetation;**
 - **Medium vegetation;**
 - **High vegetation;**
 - **Water;**
 - **Land;**
 - **Buildings, bridges;**
 - **Cars, low buildings.**

6. [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.

7. [optional] Set **Open Dem after filtering** checkbox to open output DEM in a new layer immediately after filtration completion.
8. Click OK. The process of saving a new DEM starts.

7.6. Transformation of DEM coordinates

7.6.1. DEM transpose

The system provides opportunity to transpose DEM from the left coordinate system to the right one and vice versa. To do this use the **Terrain > DEM > Transpose** menu item. Such a transformation results in swap of DEM rows and columns.

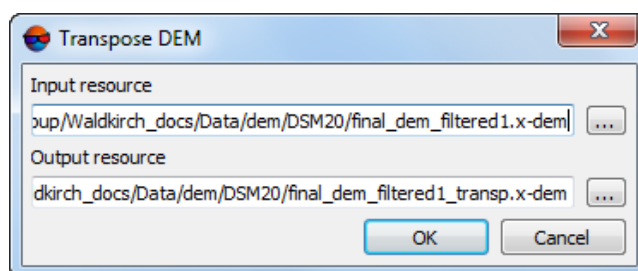


Fig. 152. The DEM transpose window



This feature is used, for example, if vector objects coordinates are incorrectly specified (see the chapter “Transformation of objects coordinates” in the “[Vectorization](#)” User Manual).

7.6.2. Transformation of DEM coordinate system

The system provides opportunity to transform DEM from one coordinate system to another.

In order to transform DEM to another coordinate system perform the following actions:

1. Select **Terrain > DEM > Transform to another coordinate system**. The **Conversion DEM to another coordinate system** window opens.

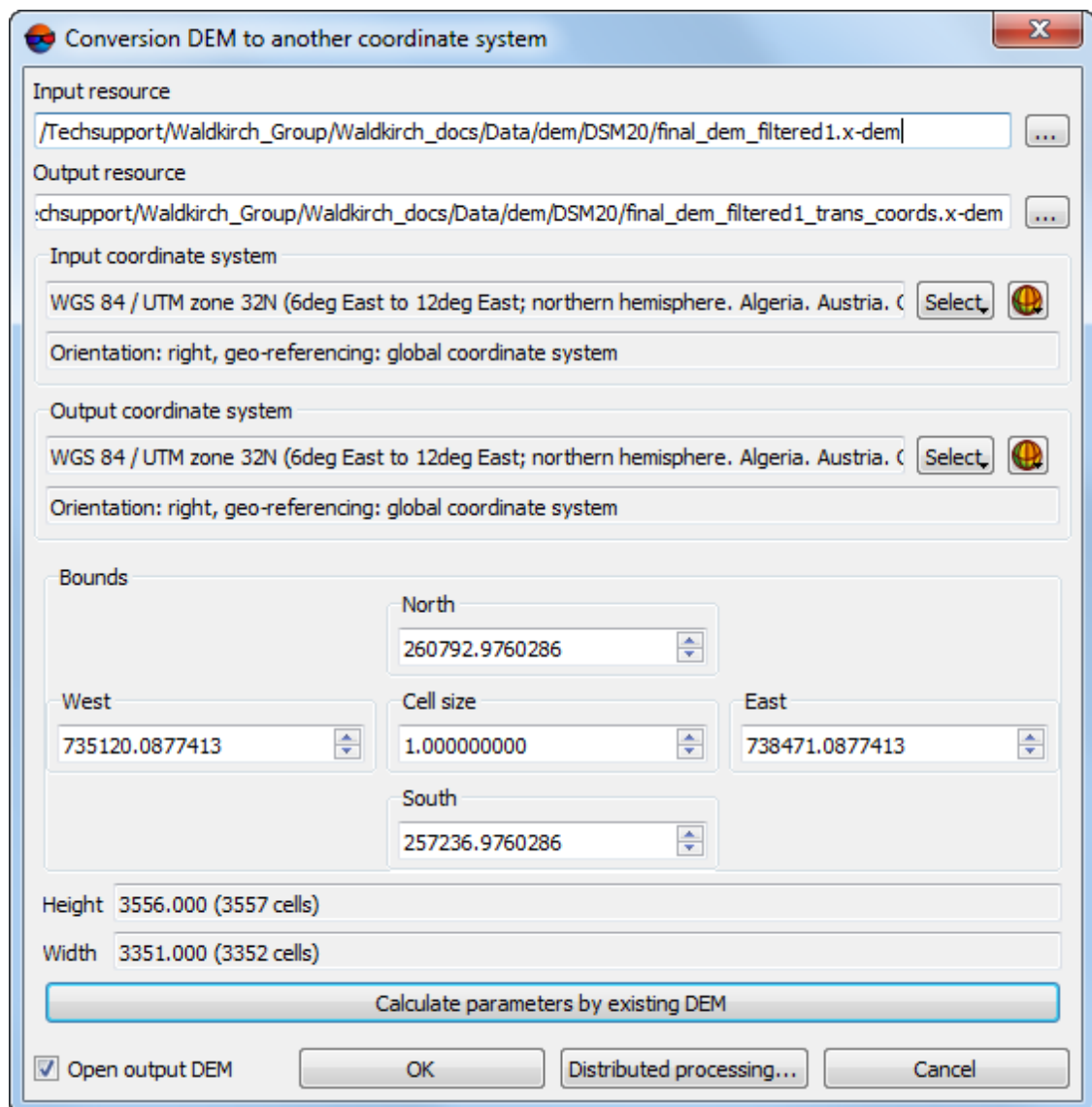




Fig. 153. Transform DEM to another coordinate system

2. Click the  button in the **Input DEM** section and specify a source DEM file in active profile resources.
3. Click the  button in the **Output DEM** section and specify DEM name and target folder in active profile resources.

4. In the **Initial coordinates system** section select actual Coordinate system of DEM (if it is different from project coordinate system). Perform the following actions to do this:

1. Click the **Select...** button to specify initial coordinate system.

Coordinate system is specified using one of the following ways:

- **From DB** – from international or Russian coordinate system database (see 'Coordinate systems databases' in the [Project creation](#) User Manual);
- **From file** – allows to select coordinate system from files with *.x-ref-system, extension located out of active profile resources;
- **From resource** – from files with *.x-ref-system extension located in active profile resources, for example, to select coordinate system from another active profile project.
- **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).



The system also allows to select coordinate system from a list of recently used coordinate systems.

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



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

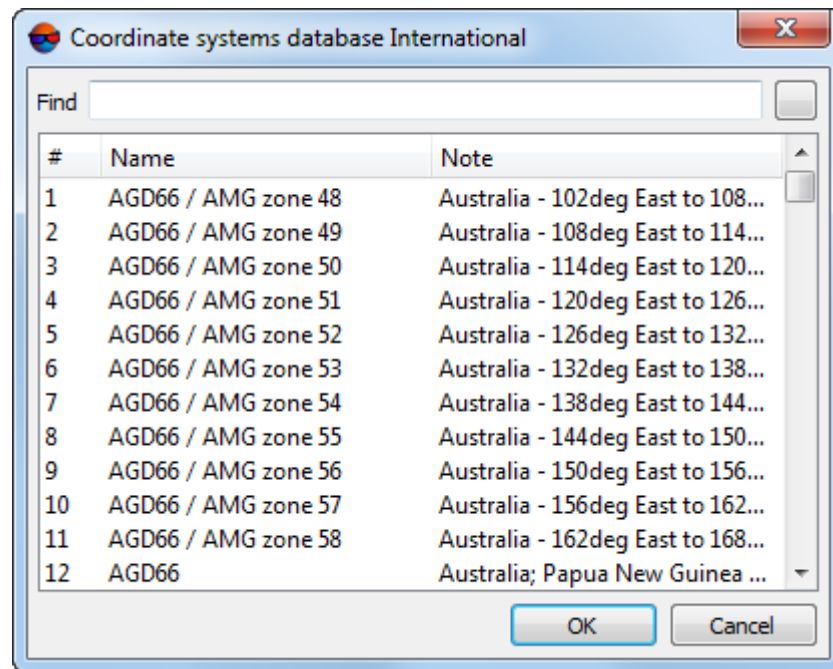



Fig. 154. Window used to select coordinate system from coordinate system database

3. [optional] 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.

5. In the **Output coordinate system** define target coordinate system in which it is necessary to convert the DEM. To do this perform actions from step 4.
6. 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, as well as number of DEM nodes.
7. Specify **Cell size** in meters to change size of output DEM cell.

8. [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.
9. [optional] To open DEM after recalculation of coordinate system the **Open output DEM** checkbox is set on by default. Set the checkbox off to not load the created file in the project.
10. Click OK to convert DEM coordinate system.

To use distributed computing for changing of DEM coordinate system, 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.

7.6.3. Batch transformation of DEMs coordinate system

The system provides opportunity to transform multiple DEMs from one coordinate system to another.

To perform this do the following:

1. Select **Terrain > DEM > Batch transform to another coordinate system**. The **Group DEM transform** window opens.

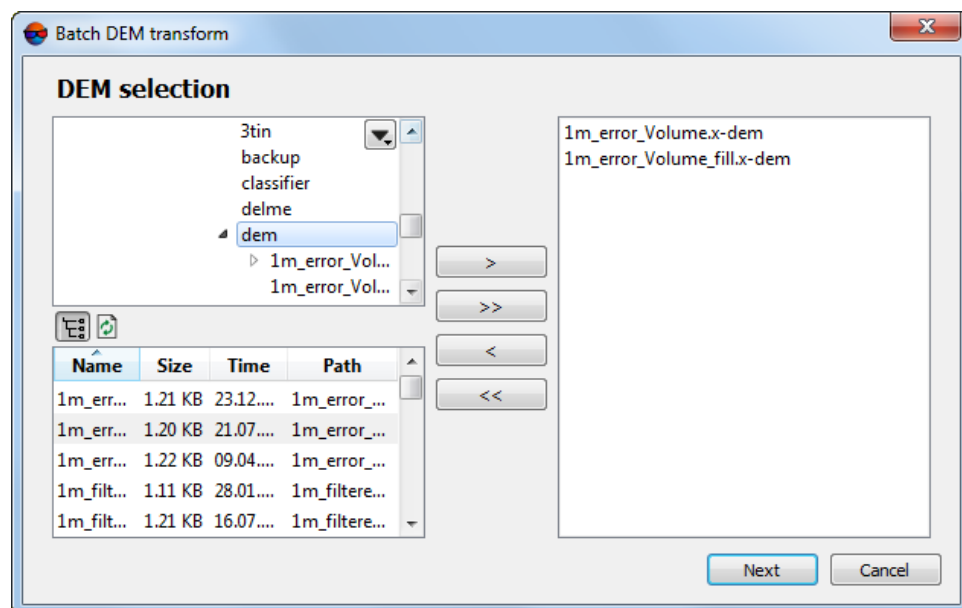









Fig. 155. The Group DEM transform window





2. Choose a folder containing DEMs in resources tree.

  button allows to display all available resources in nested files.

  button allows to refresh part of the window with resources.

  button allows to show a list with 10 recently selected resources.

3. Select file with DEM in the list and click the  button to add the file.

  and  allow to add to the list or remove from it all added files with vector objects, the  button allows to remove selected file from the list.

4. Repeat the 2 and 3 actions to add next files with DEMs.

5. Click the **Next** button.

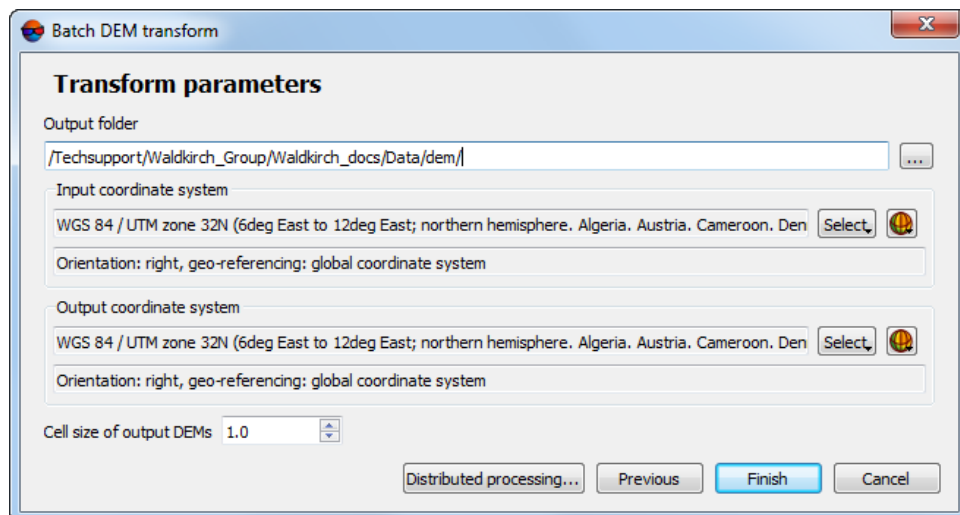



Fig. 156. Output folder and coordinate system settings window

6. Click the  button in the **Output folder** section and select a folder in active profile resources.
7. In the **Input coordinates system** section select actual Coordinate system of DEMs (if it is different from project coordinate system). Perform the following actions to do this:

1. Click the **Select...** button to specify initial coordinate system.

Coordinate system is specified using one of the following ways:

- **From DB** – from international or Russian coordinate system database (see 'Coordinate systems databases' in the [Project creation](#) User Manual);
- **From file** – allows to select coordinate system from files with *.x-ref-system, extension located out of active profile resources;
- **From resource** – from files with *.x-ref-system extension located in active profile resources, for example, to select coordinate system from another active profile project.
- **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).



The system also allows to select coordinate system from a list of recently used coordinate systems.

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



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

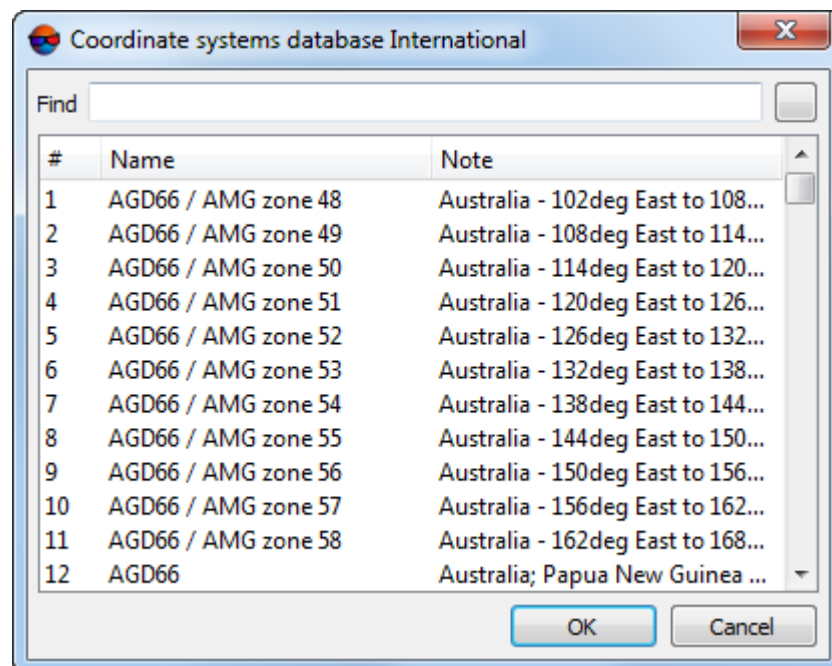



Fig. 157. Window used to select coordinate system from coordinate system database

3. [optional] 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.

8. In the **Output coordinate system** define target coordinate system in which it is necessary to convert DEMs. To do this perform actions from step 7.
9. Specify the **Cell size of output DEMs** in meters.
10. Click OK to start operation of batch DEMs transformation.

In order to perform batch DEM transformation using distributed processing, 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.

7.6.4. Rebuilding DEM considering last adjustment

The system provides opportunity to rebuild DEM considering last adjustment (for example, if the project was copied and adjusted after DEM creation once more).

To do this perform the following actions:

1. Select **Terrain › DEM › Rebuild considering last adjustment**. The **Rebuild considering last adjustment** window opens.

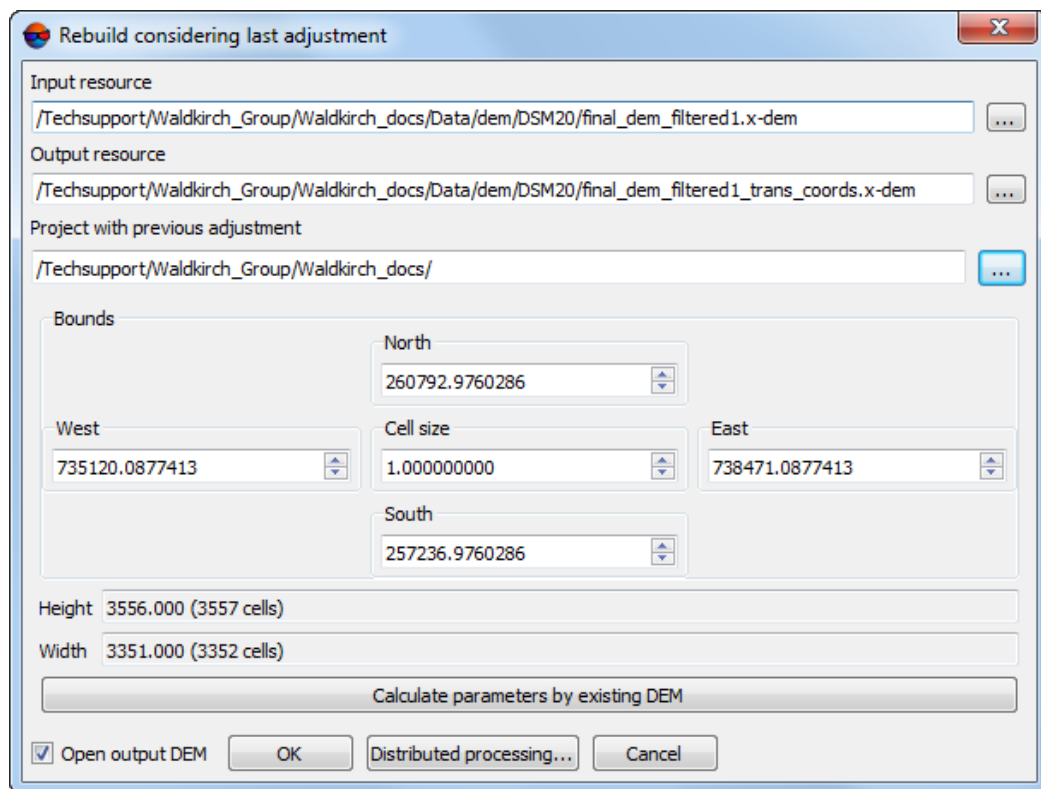





Fig. 158. The Rebuild considering last adjustment window

2. Click the  button in the **Input resource** section and specify a source DEM file in active profile resources.
3. Click the  button in the **Output resource** section and specify DEM name and target folder in active profile resources.
4. Click the  button in the **Project with previous adjustment** section and select the project with previous adjustment in the **PHOTOMOD projects** window;
5. 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, as well as number of DEM nodes.
6. Specify **Cell size** in meters to change size of output DEM cell.
7. [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.
8. [optional] To open DEM after rebuilding the **Open output DEM** checkbox is set on by default. Set the checkbox off to not load the created file in the project.

9. Click OK to rebuild DEM.

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.

7.7. Null cells in DEM

7.7.1. General information

During automatic creation of regular relief model (DEM) some part of the cells can not be defined. Such DEM cells ('gaps'), which elevation is not known, are called *null cells*.

The system provides possibility to restore null cells of DEM using the following methods of null cells filling:

Table 8. Brief description of “Fill NULL cells” menu

Menu items	Function
linear interpolation	see Section 7.7.2
smooth interpolation	see Section 7.7.3
By adjacent value	see Section 7.7.4
By constant value	see Section 7.7.5
By minimum value	see Section 7.7.6
Build convex hull	a border is created by filling the null cells in border areas in such a way that DEM has the most smooth border
Set elevation in selected polygons...	to set the same elevation of DEM cells inside and outside of selected polygons
Interpolate elevation in selected polygons...	to interpolate a value of DEM cells inside and outside of selected polygons

The system also allows to [transform DEM cells with specified elevation to null cells](#).



Filling DEM null cells is also used to change input DEM cell size.

7.7.2. Filling null cells using linear interpolation

The system provides possibility to restore null cells of DEM using the linear interpolation method. Null cells filling using this method is performed by creating TIN from the DEM and calculation of null cells elevation using method of linear interpolation by created TIN. When using this method it is possible to change the DEM borders and cell size, as well as to remove 'sharp' peaks of TIN.

To fill DEM null cells by linear interpolation method perform the following methods:

1. Select **Terrain > DEM > Fill null cells > Linear interpolation...** The **Parameters of filling null cells in DEM by linear interpolation** window opens.

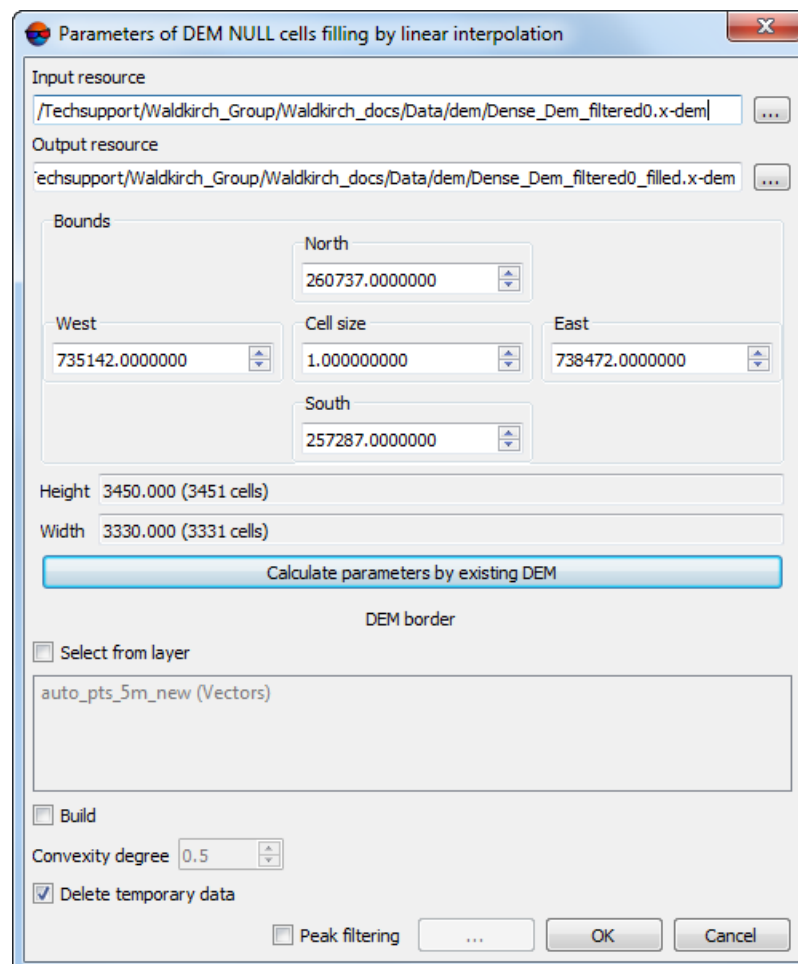


Fig. 159. Parameters of filling null cells using linear interpolation

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>_corr.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. In the **Extents** section specify DEM boundaries in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters, as well as number of DEM nodes.
5. Specify **Cell size** in meters to change size of output DEM cell.
6. [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.
7. [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;

8. [optional] In the **DEM border** section, set the **Build** checkbox to create borders of the output DEM automatically;
 - Set the **convexity degree**.



The high **convexity degree** means that the DEM border is created by filling the null cells in border areas in such a way, that DEM has the most smooth border.

9. [optional] To remove processing data after the calculation is completed set the **Delete temporary data** checkbox on;
10. [optional] To remove sharp peaks in DEM cells with filled elevation values set the **Peak filtering** checkbox on and click the  button. The **Parameters** window opens.

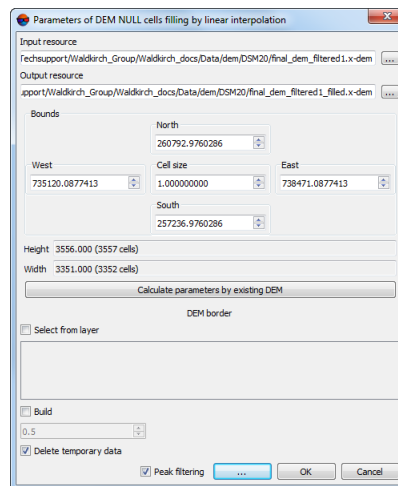


Fig. 160. Parameters of peak filtering

Input the following peak filtering parameters:

- **Edge length** – allows to specify maximal edge length in DEM cell (in meters);
- **Elevation shift up more than** – allows to specify maximal deviation of cell by elevation above adjacent cells (in meters).

11. Click OK. The system starts calculation of null cells elevation using linear interpolation method. After that the system creates new DEM layer in the *Manager* and shows a message about operation successful or unsuccessful completion.

7.7.3. 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....** The **Parameters of filling null cells in DEM** window opens.

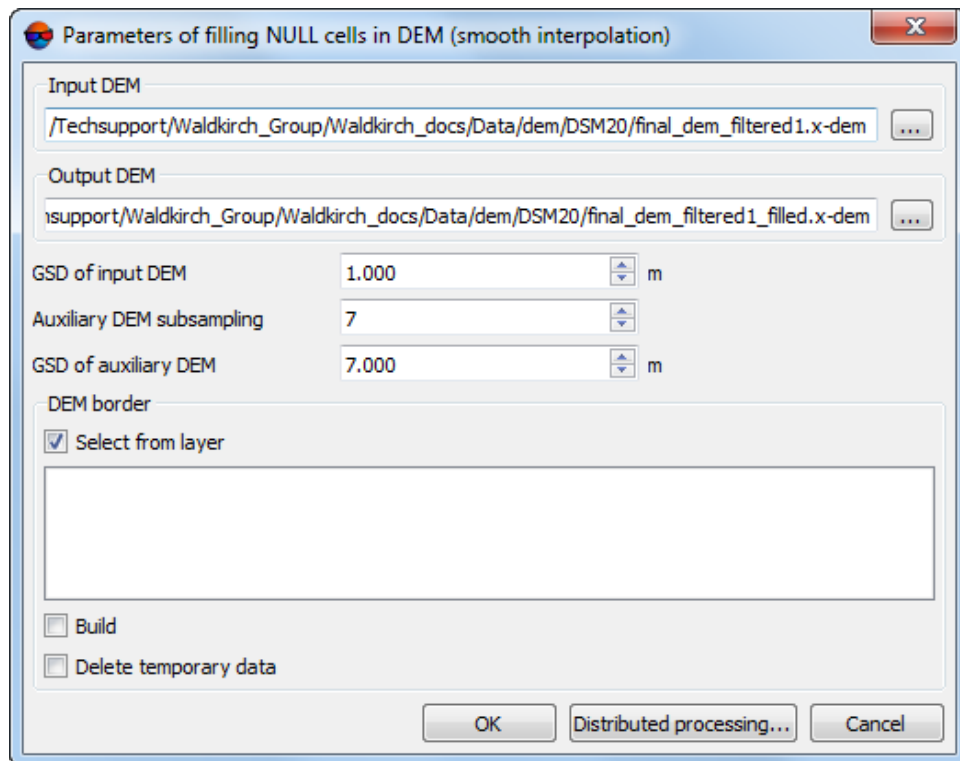




Fig. 161. Parameters of filling null cells using smooth interpolation

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.

7.7.4. Filling null cells by adjacent value

The method of filling the null cells using closest value is based on an analysis of the values of cells surrounding the null cell. This operation allows to fill DEM null cells by a closest or average value of non-null cells for a given neighborhood. The output DEM is the same in size and dimensions with the original, or is a thinned source DEM with the degree of thinning divisible by 2 (depending on the **Processed pyramid level** parameter value).

To restore a DEM with null cells using this method perform the following actions:

1. Select **Terrain > DEM > Fill null cells > By adjacent values**. The **Parameters of DEM null cells filling by adjacent values** window opens.

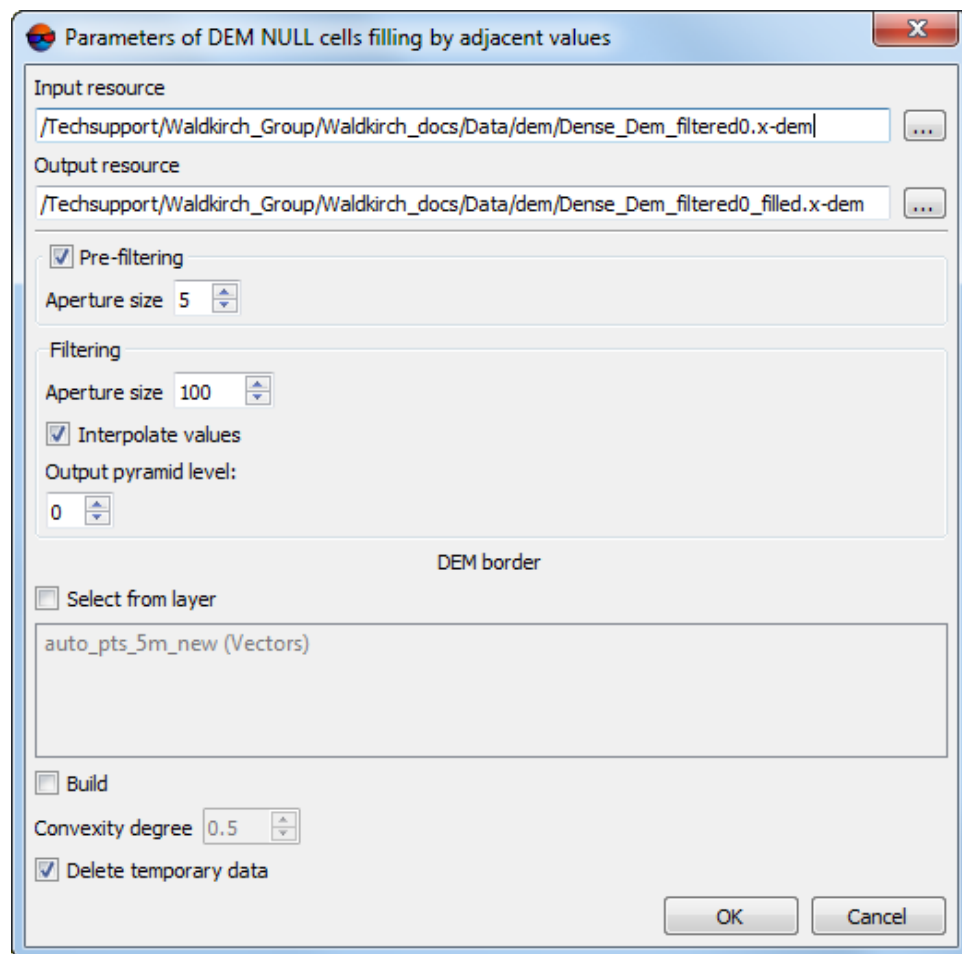



Fig. 162. Parameters of DEM null cells restoring

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. Specify a path for **output DEM** in active profile resources.



By default the system suggests the *<input dem name>_corr.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. [optional] **Pre-filtering** – is an optional step, which allows you to fill small areas of null cells (smaller than a given neighborhood area). It is applied to the main layer of the source DEM.

5. [optional] If you decided to perform the pre-filtering, specify the **Aperture size** value. The value defines a size of a neighborhood, within which the height values to fill the null cells are searched. Recommended aperture size is 3-10.
6. The main step is performed on selected pyramid level of output DEM. In the **Filtering** section specify the following parameters of the main step:

- **Aperture size** – defines a size of a neighborhood, within which the height values to fill the null cells are searched. Required value depends on the size of null cells clusters located on the source DEM and the level of used pyramid.



It is recommended to specify the aperture size value several times larger than for the preliminary stage.

- **Interpolate values** – when the checkbox is on the null cell is filled by the average height value, taken in specified neighbourhood, if the checkbox is cleared – by closest (geometrically) height value on the specified neighbourhood.



To calculate the average heights value the system uses bilinear interpolation. This improves an output DEM quality, but increases processing time.

- **Output pyramid level** – allows to select a pyramid level of the source DEM used to create an output DEM. If the source DEM is too detailed, select the pyramid level value more than 0 to use thinned image.

7. [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;

8. [optional] In the **DEM border** section, set the **Build** checkbox to create borders of the output DEM automatically;

- Set the **convexity degree**.



The high **convexity degree** means that the DEM border is created by filling the null cells in border areas in such a way, that DEM has the most smooth border.

9. [optional] To remove processing data after the calculation is completed set the **Delete temporary data** checkbox on;
10. Click OK. Null cells of a DEM are filled considering the specified parameters.

7.7.5. Filling null cells by constant value

The system provides possibility to fill null cells of DEM by constant value.

To fill DEM null cells with constant value perform the following methods:

1. Select **Terrain > DEM > Fill null cells > By constant value....** The **Fill DEM null cells by constant value parameters** window opens.

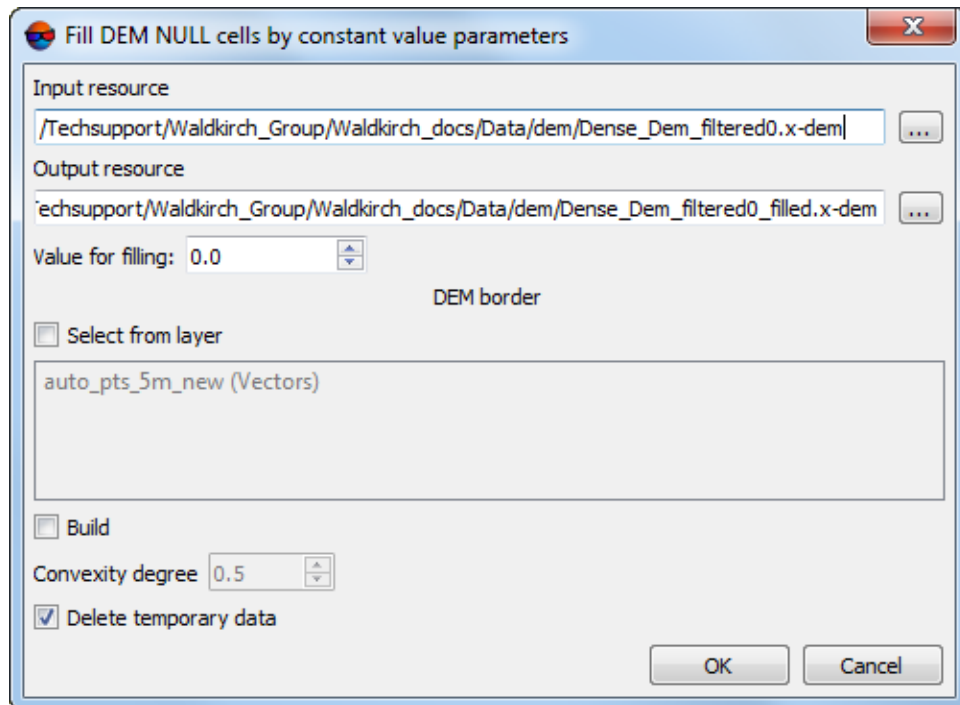




Fig. 163. Parameters of filling null cells of DEM by constant value

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 **Value for filling** of null cells.

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;

- Set the **convexity degree**.



The high **convexity degree** means that the DEM border is created by filling the null cells in border areas in such a way, that DEM has the most smooth border.

7. [optional] To remove processing data after the calculation is completed set the **Delete temporary data** checkbox on;
8. Click OK. The system starts filling of null cells operation. After that the system displays a message about number of changed cells and creates new DEM layer in the *Manager*.

7.7.6. Filling null cells by minimum values

The system provides possibility to fill null cells of DEM by minimum values.



The system also provides possibility to interpolate areas without data on orthophotomap (true ortho) [created together with the DEM](#).

The method of filling the null cells using minimum values is based on an analysis of the values of cells surrounding the null cell. This method allows to exclude nearby buildings from interpolation of DEM null cells (see the figure below, paragraph 11).

To fill DEM null cells with minimum values perform the following:

1. Select **Terrain › DEM › Fill null cells › By minimum values....** The **Fill NULL cells** window opens.

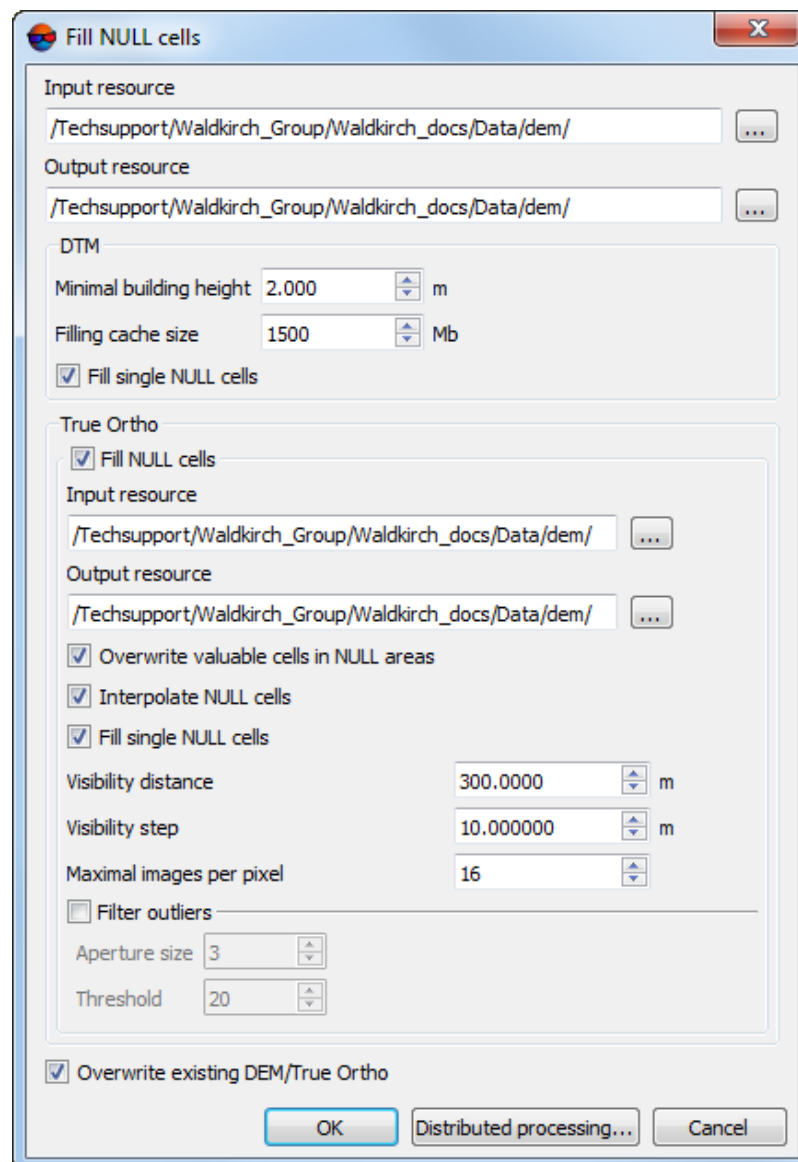






Fig. 164. Parameters of filling null cells of DEM by minimum values

2. In the **DEM** section click the  button to select input DEM in active profile resources.
3. [optional] To define name and target folder of output DEM in active profile resources, click the  button.
4. In **DEM** section, set **Minimum building height** in meters to exclude buildings from interpolation of DEM null cells (see the figure below);
5. In **DEM** section, set **Cache size** when filling option in megabytes – maximum cache size allocated for a single task when creating true ortho;

6. [optional] In **DEM** section, clear the **Fill single NULL cells** checkbox in order not to interpolate single DEM NULL cells;
7. [optional] In **True Ortho** section, clear the **Fill NULL cells** checkbox in order not to interpolate unfilled areas on orthophotomap;
8. In the **True Ortho** section click the  button to select input true ortho in active profile resources.



For correct work of interpolation algorithm it is required to select the orthophotomap that was **created together with the DEM** in the same boundaries (see step 2).

9. [optional] To define name and target folder of output true ortho in active profile resources, click the  button.
10. [optional] Set the following parameters of unfilled areas interpolation on orthophotomap:

- **Overwrite valuable cells in NULL areas** – allows to exclude *isolated DEM areas* from interpolation of unfilled true ortho areas;



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 orthophotomap areas;
 - **Fill single NULL cells** – allows to interpolate areas without data having one pixel size on 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;
 - **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;
 - **Filter outliers** – allows to perform orthophotomap median filtering with preset **Threshold**. The statistics is collected in the neighborhood whose area is determined by the **Aperture size**. This filter is used for removing single pixels with unnatural brightness.
11. [optional] set the **Overwrite existing DTM/TrueOrtho** checkbox in order not to use intermediate data from the working folder but overwrite them.

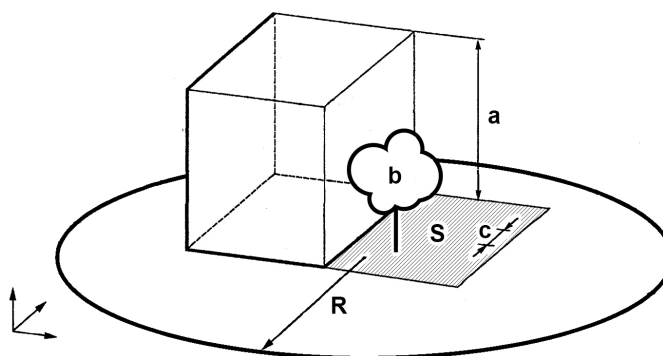


Fig. 165. An example of unfilled area on true ortho, where **a** is the building height (see **Minimum building height** parameter), **b** is the “isolated DEM area” among DEM NULL cells (see **Overwrite valuable cells in NULL areas** checkbox), **c** is the **Visibility step**, **R** is the **Visibility 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).

12. Click OK. The system starts filling of null cells operation. After that the system displays a message about number of changed cells and creates new DEM layer in the *Manager*.

To use distributed processing for DEM null cells filling, 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.

7.7.7. Converting cells to null

The system allows to convert DEM cells with specified elevation to null cells.



It is recommended to apply this method if there are errors after importing DEM.

In order to convert cells to null ones perform the following actions:

1. Select **Terrain > DEM > Filter > Cut DEM by Z-range...**. The **Parameters of converting DEM cells to null values** window opens.

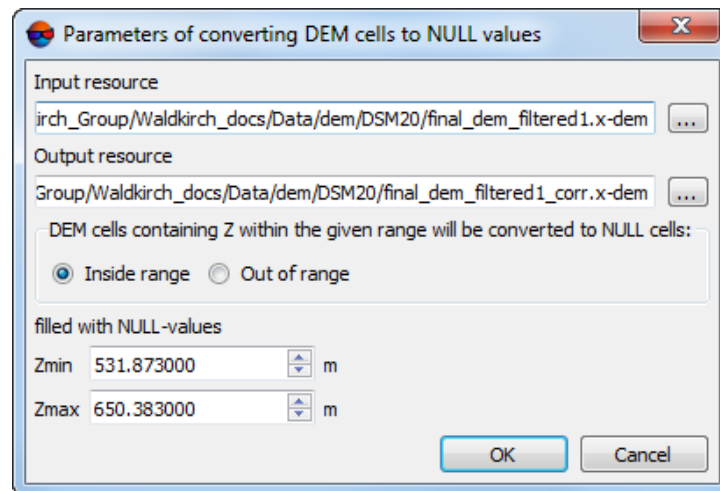




Fig. 166. Parameters of converting DEM cells to null values

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>_corr.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. Specify a range of DEM cells to be converted to null in the **Zmin** and **Zmax** fields in meters.



The system allows to convert DEM cells, located both **inside the range** and **outside the range**.

5. Click OK. After that all output DEM cells, which elevation falls within specified Z-range are converted into null cells. After the operation is completed, an informational message about the number of deleted cells is displayed.

7.8. DEM editing

7.8.1. DEM merging

The system provides possibility of merging (joining) of adjacent DEMs that have overlap.

To merge adjacent DEMs perform the following actions:

1. Select **Terrain > DEM > Merge DEMs....** The **DEM merging – Step 1 from 2:** window opens **Source data**.

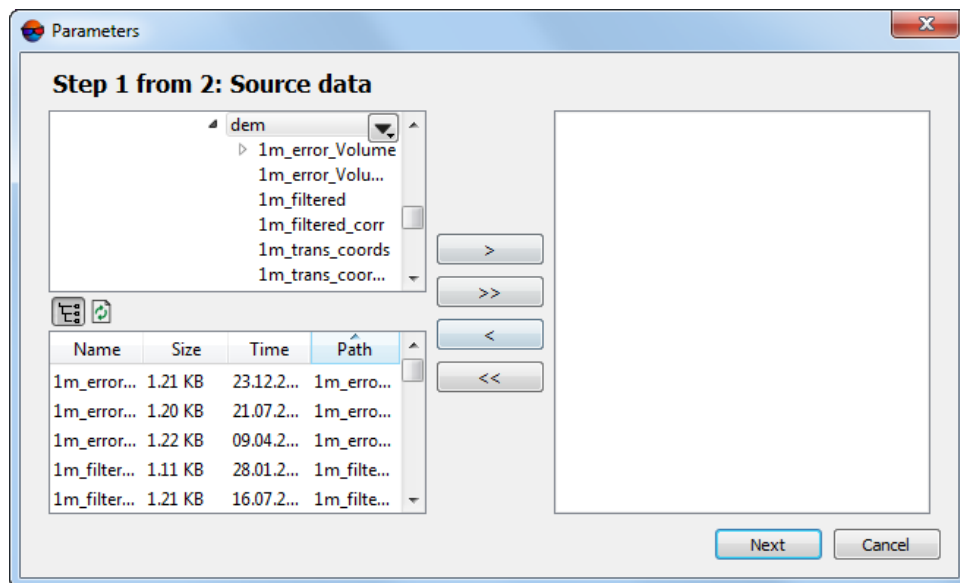




Fig. 167. Selection DEMs for merging


2. Choose a folder containing DEM in resources tree.






The  button allows to display all available resources in nested files. The  button allows to refresh part of the window with resources.



The  button allows to show a list with 10 recently selected resources.

3. Select the DEM that has overlap with the adjacent one in the list, and click the  button to add the DEM to be merged.



The  and  buttons allow to add to the list or remove from it all added DEMs, the  button allows to remove selected DEM from the list.

4. Repeat the 2 and 3 actions to add next DEMs.



It is necessary to choose for merging at least two files containing DEMs with overlap area.

5. Click the **Next** button. The **DEM merging – Step 2 from 2: Parameters** window opens.

Parameters

Step 2 from 2: Parameters

Bounds

North: 260616.9140000

West: 736097.2370000

Cell size: 1.000000000

East: 738179.2370000

South: 257388.9140000

Height: 3228.000 (3229 cells)

Width: 2082.000 (2083 cells)

Calculate parameters by existing DEM

Inside overlapping area, use:

☒ Top DEM ☐ Bottom DEM ☐ Average value ☐ Weighted mean value

☒ Open output DEM

Distributed processing... Previous Finish Cancel

Fig. 168. Parameters of DEMs for merging

6. 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, as well as number of DEM nodes.
7. Specify **Cell size** in meters to change size of output DEM cell.
8. [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.
9. In the **Inside overlapping area, use** section select the DEM, which values should be used in the overlap area:
 - **Top DEM** – use data from top DEM (the first or the top DEM in the list, added for merging, in the **DEM merging – Step 1 from 2: Source data** window);
 - **Bottom DEM** – use data from bottom DEM (the second or the bottom DEM in the list, added for merging, in the **DEM merging – Step 1 from 2: Source data** window);
 - **Average value** – calculate and use average cells values of adjacent DEMs in overlap area.
 - **Weighted mean value** – while calculating average values in overlapping areas to take into account an estimation of errors in DEM heights with the subsequent filtering “rough” ones;



It is recommended to use **Weighted mean value** for calculating DEM value in overlapped area if DEMs being merged can contain “rough” errors.

Fig. 169. Calculating heights inside overlapping area with weighted mean value

Setup the following parameters of calculating weighted mean values inside overlapping area:

- **Apriory DEM accuracy by Z** is the RMS limit involved in calculating the permissible deviation from the average value for overlapping DEM cells. **Apriory DEM accuracy by Z** determines the strictness of rejection of overlapping DEMs whose data are used for calculation of the average value in the current cell and is set by the user based on the available information about the nature of the data used.

As a result of rejection, the resulting cell of the output DEM can have either the average height of the cells of all DEMs overlapping at a given point (except the rejected values) or be the so-called hole.

In certain cases, when using DEMs created in *PHOTOMOD*, the **Apriory DEM accuracy by Z** can be pre-estimated by the user, at least for some part of the input data.

DEMs **built by SGM method** and no having any subsequent additional processing contain data of priori accuracy estimation.

Priori accuracy estimation is calculated based on image GSD and intersection angle of stereopair on which DEM is built.

To find data of priori accuracy estimation select **Service > Explorer**. **PHOTOMOD Explorer** window is opened. At the left side of the window in the folder containing DEM open subfolder "*Temp/Dems*", which contains DEMs with *.x-dem extension. They are built for each stereopair separately.

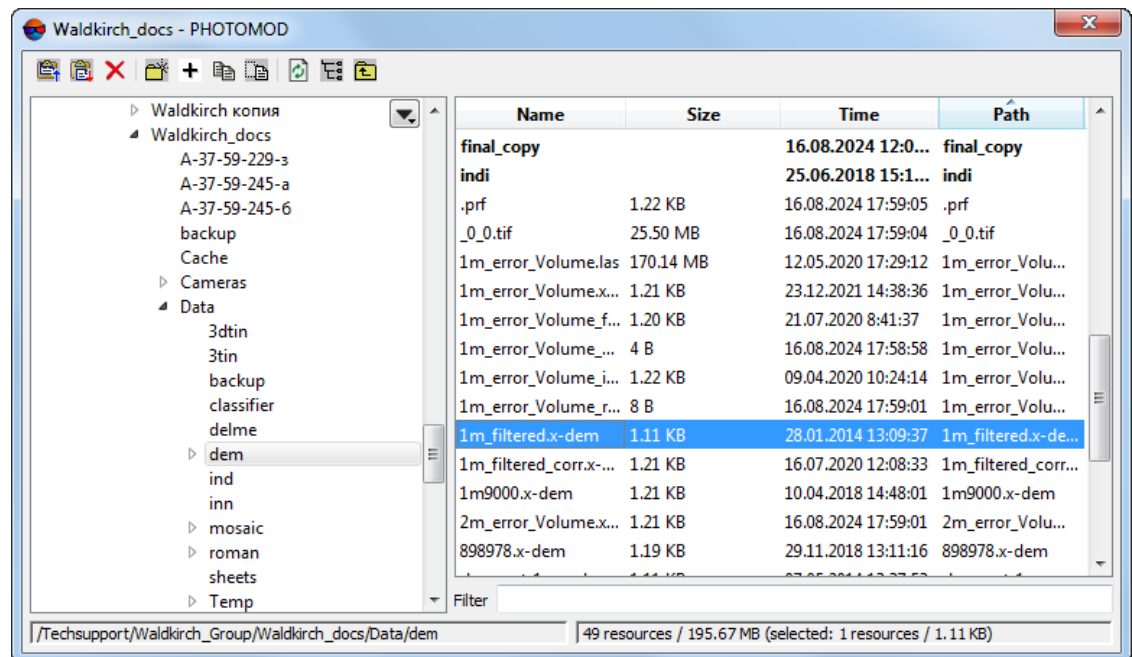


Fig. 170. PHOTOMOD Explorer window

At the right side of **PHOTOMOD Explorer** window open a context menu with a click of mouse right button in the file name of any DEM generated for separate stereopair. Select **View as text**. DEM file is opened in a text editor.

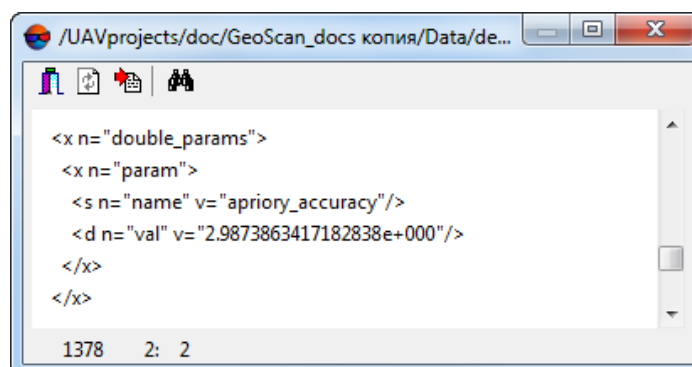


Fig. 171. DEM file opened in a text editor

Information on priori accuracy estimation is contained after line `<s n="name" v="apriory_accuracy"/>`, where `v` is a desired value. I.e. `<d n="val" v="2.9873863417182838e+000"/>`.

- **Minimum number of counts in overlap area** allows to setup a reliability of output DEM cell values. Given parameter defines a minimal number of values needed for calculating final value of output DEM cell. If **Minimum number of counts in overlap area** is more than 1, all the DEMs cells placed in areas without DEMs overlaps **will not be recorded**.

10. [optional] By default the **Open output DEM** checkbox is set on that allows to open the out put DEM after the operation is completed. Set the checkbox off to not load the resulting DEM in the project.
11. Click OK to start DEM merging process.

To use distributed computing for DEMs merging 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.

7.8.2. Change DEM parts elevation

The system allows to align selected part of DEM by elevation. In this case, a fixed elevation value is assigned to all cells of the DEM part outlined by selected polygon.

To align DEM part by elevation perform the following actions:

1. Create or load a vector layer (see the “[Vectorization](#)” User Manual).
2. Create single or multiple polygons, that outline selected parts of the DEM, or load polygons layer to use them as boundaries.
3. Select at least one polygon that will be used as a boundary.
4. Choose **Terrain › DEM › Fill NULL cells › Set elevation in selected polygons**. The **Parameters of DEM elevation assigning in polygons** window opens.

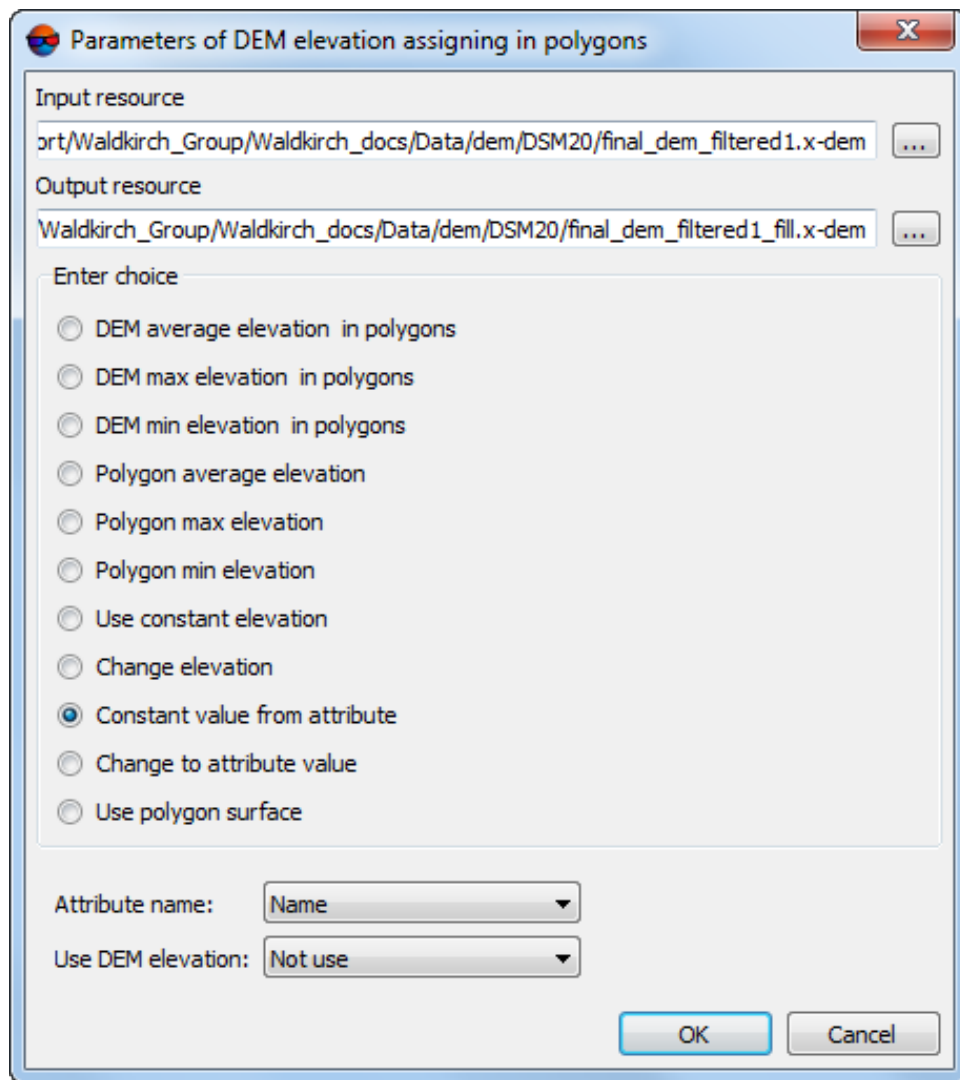




Fig. 172. Parameters of DEM elevation assigning in polygons

5. 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.

6. [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>_fill.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

7. Specify value of DEM cells in each selected polygon:

- **DEM average elevation in polygons;**
- **DEM max elevation in polygons;**
- **DEM min elevation in polygons;**
- **polygon average elevation;**
- **polygon max elevation;**
- **polygon min elevation;**
- **Use constant elevation;**



Set the **Constant elevation** in meters.

- **Change elevation** – allows to **shift** selected part of DEM by Z axe in both directions.
- **use polygon surface** – allows to build a TIN-surface created using the same polygon into the DEM (within a polygon).



This function allows to build an object (e.g. a building) described by the abovementioned vector polygon into the DEM. At the same time, to calculate new values of DEM cells within a polygon, only the TIN surface built for this (based on the same polygon) is used (without interpolation of old DEM values).

8. Click OK. After that a specified elevation values is assigned to all DEM cells inside selected polygons.

7.8.3. DEM shift

The system provides possibility to shift the whole DEM by one or more axes using parallel displacement.

In order to shift a DEM perform the following actions:

1. **Load** or **create** a DEM;
2. Select **Terrain › DEM › Shift**. The **DEM shift** window opens;

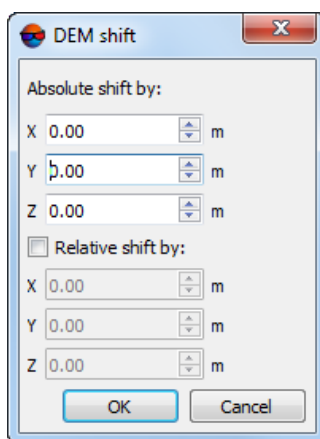


Fig. 173. Parameters of DEM shift

3. Specify the value of the **Absolute shift by** axes in the **X, Y, Z** fields in meters.



It is possible to shift a DEM by one or all axes at once.

4. Click OK. After that all points of active layer DEM are shifted on specified value by selected axes with help of parallel displacement.

In order to shift a DEM *once again* perform the following actions:

1. Select **Terrain > DEM > Shift**. The **DEM shift** window opens. In the **Absolute shift by** section, the DEM shift will be displayed relative to its initial position.



The information on DEM's shift relative to its initial position is saved automatically after each shift operation and is available after closing the DEM layer and/or restart.

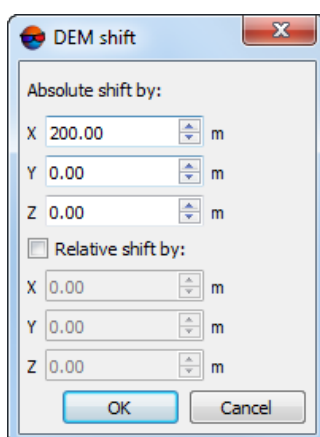


Fig. 174. Parameters of DEM shift. The DEM was shifted to 200 meters by the X-axis

2. To specify a next shift parameters perform one of the following actions:

- [optional] Change the value of the **Absolute shift by** axes in the **X, Y, Z** fields;
- [optional] Specify the value of the **Relative shift by** axes in the **X, Y, Z** fields in meters (to do this set an appropriate checkbox). In the **Absolute shift by** section, the DEM shift value(s) will be changed automatically.

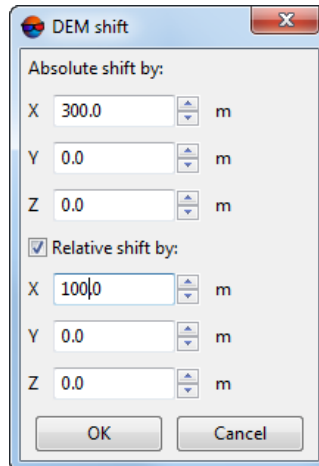


Fig. 175. Parameters of DEM shift. The DEM shift will increase by 100 meters by the X-axis. The absolute shift by the X-axis will be 300 meters

3. Click OK. After that all points of active layer DEM are shifted on specified value of the relative shift by selected axes with help of parallel displacement.

In order to restore DEM to initial state perform the following actions:

1. Select **Terrain > DEM > Shift**. The **DEM shift** window opens. In the **Absolute shift by** section, the DEM shift will be displayed relative to its initial position.

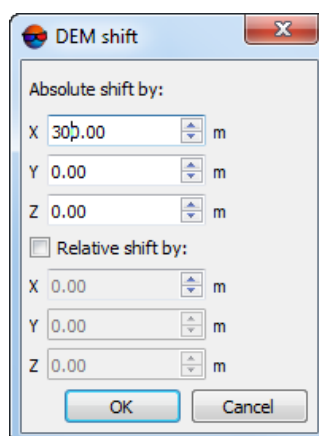


Fig. 176. Parameters of DEM shift. The DEM was shifted to 300 meters by the X-axis, during the previous actions

2. Set the zero values of the **Absolute shift by** axes in the **X, Y, Z** fields;

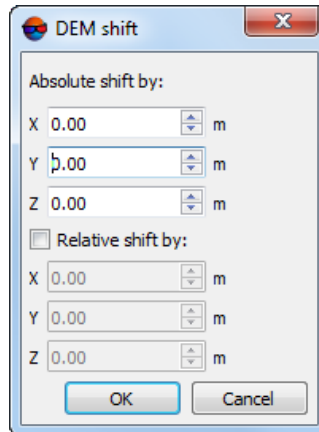


Fig. 177. Parameters of DEM shift

3. Click OK. After that all points of active layer DEM are shifted on the initial position.

7.8.4. Split DEM into sheets

Splitting of DEM into sheet is available in the system. It is used to save DEM in parts in separate files. Sheets are created inside the vector polygons.



See also the “Generators of splitting into sheets” chapter in “[Orthophotomaps creation](#)” User Manual and the “Point cloud cutting by polygons” chapter in “[LIDAR Data processing](#)” User Manual.

To split DEM into sheets perform the following:

1. [Load](#) or [create](#) DEM;
2. Load or create vector polygons to use as sheets;
3. Make sure, that the polygons used as boundaries have the joint additional *text* attribute with the unique values for the every polygon.

Name of output DEM fragments are to be set automatically from the unique attribute value of the polygon used as boundary for cutting.



The use of numeric attributes (float, integer) is not allowed for this operation.



Every polygon must have the unique value.



If there is no chosen attribute for some sheets, these sheets can be passed or continuous numbering can be used as file names.

4. Make *active* the DEM layer.
5. Choose **Terrain > DEM > Split DEM by sheets**. The **Split DEM into sheets** window opens.

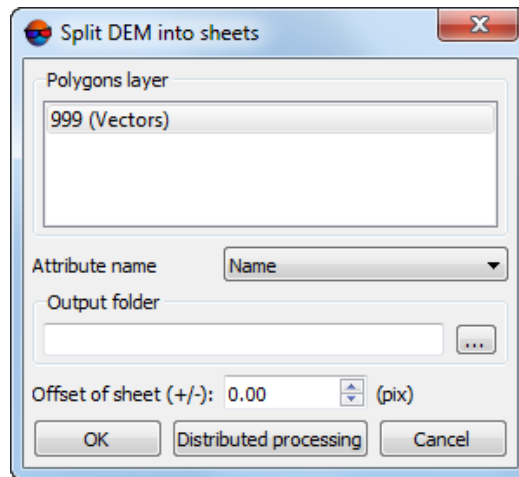


Fig. 178. The Split DEM into sheets window

6. **Choose layer with polygons** with a single left mouse click;
7. Choose the **Attribute name** to use its values as filenames.
8. Choose the **Output folder** for files, in active profile resources;
9. [optional] In the **Offset off sheet (+/-)** input field, enter a nonzero value, in pixels, to create sheet margins:
 - If the entered value is positive, the sheet is built with the given offset beyond the border of the relevant polygon;
 - If the value of the offset from the borders is negative, the sheet is built with an offset inward the polygon.
10. Click OK to start the process.

Perform the following to use distributed processing:

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.

As a result DEM splits into parts by polygons of chosen layer. Values of selected attribute are used as filenames.

7.8.5. DEM interpolating

The system supports local alignment of elevation values of DEM part, limited by a polygon on a vector layer.

In order to align a part of DEM perform the following actions:

1. Create a vector layer (see the '[Vectorization](#)' User Manual).
2. Create single or multiple polygons, that outline selected parts of the DEM, or load polygons layer to use them as boundaries.
3. Select at least one boundary polygon to be used for interpolation of cell elevation values inside it.
4. Choose **Terrain > DEM > Fill NULL cells > Interpolate elevation in selected polygons**. The **Parameters of DEM interpolation assigning in polygons** window opens.

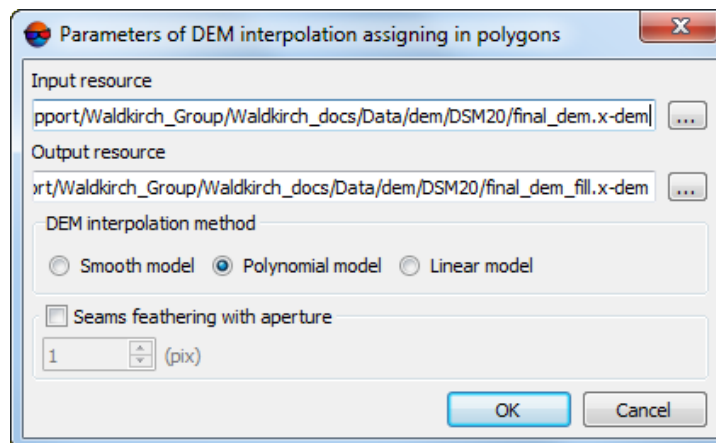




Fig. 179. Parameters of DEM interpolation assigning in polygons

5. 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.

6. [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>_fill.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

7. Specify the **DEM interpolation method**:

- **Smooth model**;
- **Polynomial model**;
- **Linear model**.

8. [optional] To smooth boundaries between unchanged parts of a DEM and its changed fragments, set the **Smooth feathering with aperture ... (pix)** checkbox, and set the aperture size in pixels;

9. Click OK. After that cell elevation values inside selected polygons (one or several) is interpolated using specified method.

7.8.6. DEM cutting by polygons

The system allows to edit the area of DEM coverage.

In order to correct DEM area, perform the following actions:

1. Create vector layer and polygons in it, or open a layer with bordering polygons (see the “[Vectorization](#)” User Manual).
2. Select one or more polygons, that will be used to cut the DEM.
3. Select **Terrain > DEM > Cut by selected polygons**. The **Polygon cutting parameters** window opens.

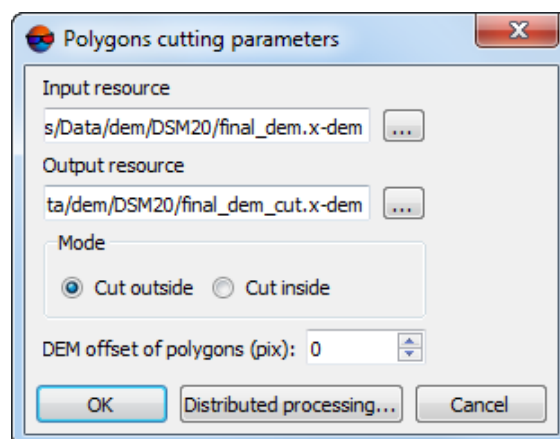




Fig. 180. Parameters of cutting by polygons

4. 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.

5. [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>_cut.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

6. Specify the **Mode** of DEM cutting:

- **Cut outside** – allows to crop DEM outside of selected polygons area;
- **Cut inside** – allows to cut DEM area located inside of selected polygons.

7. [optional] Specify **DEM offset of polygons (pix)**;

8. Click OK. After that resulting DEM obtained after editing is opened.

To edit DEM 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](#)” User Manual).
2. Click the **Distributed processing** button.

Besides, the system allows to remove edge areas of DEM, that include null cells. Perform the following actions to do this:

1. Select **Terrain > DEM > Cut null edges**. The **Cut null edges** window opens.

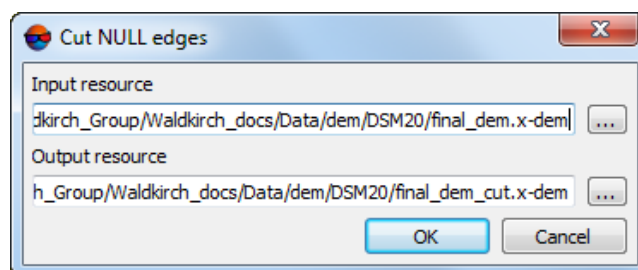




Fig. 181. DEM edges cutting

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>_cut.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. Click OK. After that the system opens a DEM without removed edges areas.

7.8.7. Convert DEM to points

The system allows to convert DEM cells points with specified simplifying.

To do this conversion perform the following actions:

1. [Load](#) or [create](#) a DEM.
2. [optional] Load vector layer and select polygons, within which it is necessary to perform the conversion.
3. Select **Terrain › DEM › Convert to points....** The **Parameters of converting to points** window opens.

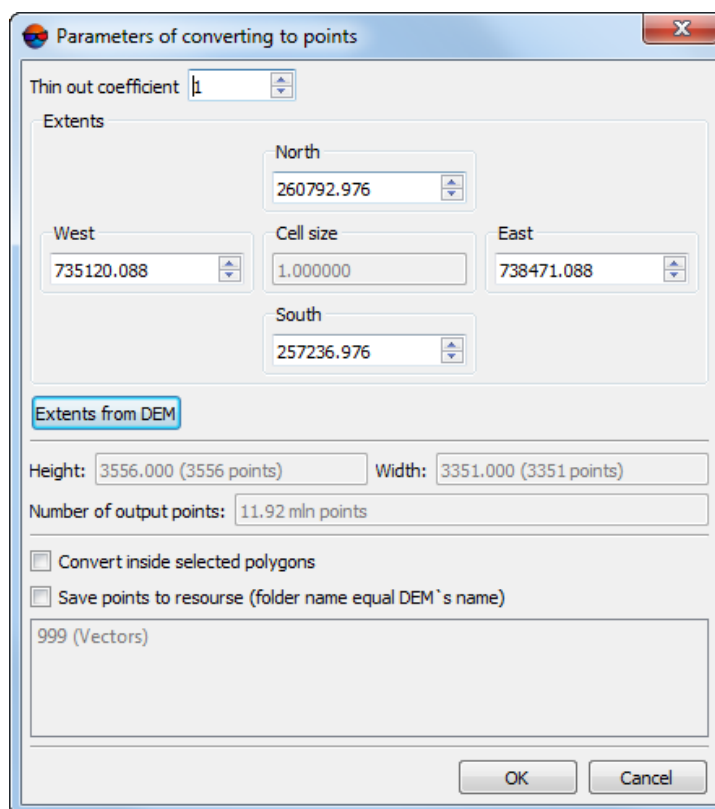


Fig. 182. Parameters of DEM converting to points

4. Specify **Thinout coefficient** of DEM.
5. [optional] Specify DEM borders in the **North**, **West**, **East**, **South** fields.

Depending on this coefficient the system calculates the **Cell size** of output DEM in meters, **Height** and **Width** in meters and in points number. Besides, the estimated **Number of output points** is displayed in this window.

The **Extents from DEM** button allows to specify values of borders of conversion area.

6. [optional] If additional vector layer with polygons is loaded, set the **Convert inside selected polygons** checkbox on and select a layer with polygons in the list to perform conversion in specified areas only.
7. Click OK. When the conversion is completed the system displays a message about number of acquired points and new vector layer with obtained points is created.



To **Save points to resource** at once automatically (without opening the created layer in the *Layer Manager*), set the appropriate checkbox.

Points will be automatically saved in the active profile resources in the folder containing the input DEM, in a separate folder with a following name: points_<DEM_name>.

7.8.8. Adding of points to DEM

The system provides possibility to add points to DEM that helps to refine DEM cells values.

Usually it is necessary to perform the following:

1. **Load** or **create** a DEM.
2. Create a vector layer.
3. Draw a polygon around the DEM, which cells values should be edited.
4. Highlight this polygon.
5. Select **Terrain > DEM > Convert to points...** to create vector layer.
6. In the **Parameters of converting to points** window select vector layer, where there is highlighted polygon.
7. [optional] Set the **rest** of parameters of DEM converting to points, if necessary.

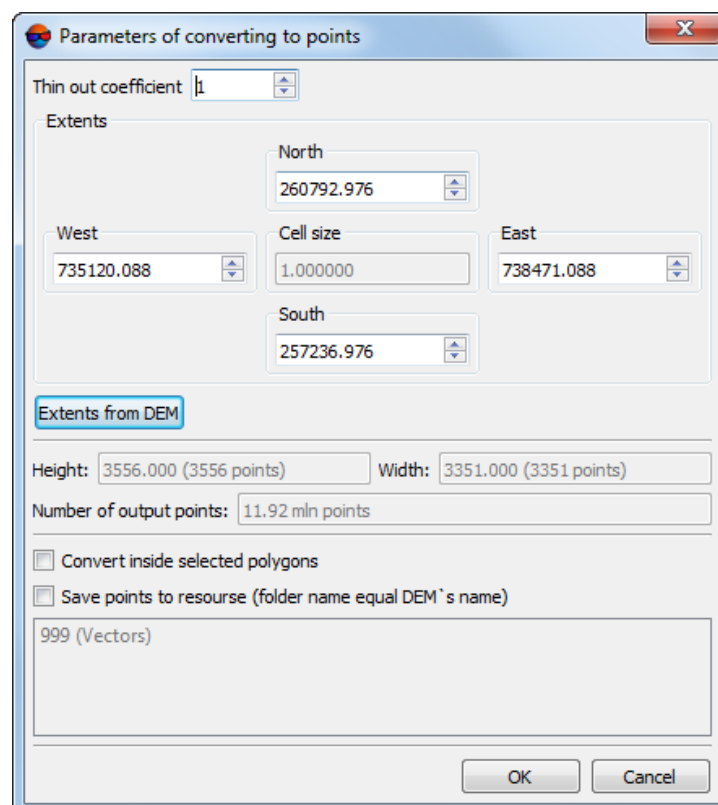


Fig. 183. Parameters of DEM conversion to points











8. Click OK. DEM area outlined by highlighted polygon will be converted to points, which are added to a new vector layer.
9. Correct location of obtained points by Z.
10. Select **Terrain > DEM > Write points to DEM**. After that the DEM is rebuild considering points layer.






It is impossible to cancel the operation. The system overwrites the active DEM *automatically*.

7.8.9. The DEM editing toolbar

The system provides possibility of fast editing of DEM areas using selecting tools (see details in the “Objects selection” chapter of the “[Vectorization](#)” User Manual). For this there is an additional toolbar **DEM editing** with buttons that allow to perform the following operations:

-  – allows to [cut DEM area](#) located inside of selected polygon;
-  – allows to [set elevation](#) of DEM cells inside of selected area;
-  – allows to interpolate selected DEM area [using smooth model](#);
-  – allows to interpolate selected DEM area [using polynomial model](#);
-  – allows to interpolate selected DEM area [using linear model](#);
-  – allows to [shift](#) the whole DEM by one or more axes using parallel displacement;
-  – allows to [add points to DEM](#) that helps to refine DEM cells values;
-  – enables the mode of [selective object filtering](#);
-  – enables the [object's preselection mode](#) (without deleting);
-  – allows to set the [selective filtering of objects](#) parameters;

The system provides for canceling some of the operations placed in the **DEM editing** toolbar. Standard functions of the action log are used for this (, , and  buttons of the main toolbar).

The undo log is described in detail in “Editing of active layer” of “[General information](#)” User Manual and its settings are given in “Settings of undo parameters” of “[General system's parameters](#)” User Manual.

A separate list of actions that can be cancelled is maintained for each layer. It is impossible to undo if too many operations are run automatically. In this case, the appropriate warning appears, and the list of executed operations is not maintained.

It is impossible to undo DEM shift (📏) and adding points (📍), and the action log is never maintained for these operations. Changes made to DEM using these two quick editing tools are immediately saved. In this case, make a DEM copy before editing (**Terrain > DEM > Save a copy...**).

If DEM shift or point adding were preceded by operations recorded in the action log (📍, 📏, 📐, ax^2 or 📐, cancellation is allowed), in this case DEM shift or point adding will interrupt the action log and cause inability to cancel previous operations. An appropriate message appears.

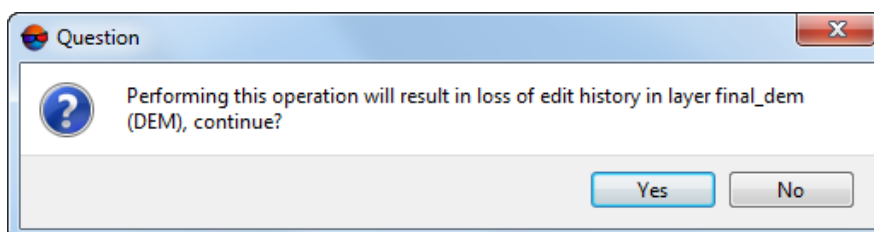


Fig. 184. Предупреждение

To quickly edit a DEM area perform the following actions:

1. **Load** a DEM or make active a layer with DEM.
2. Click the button of the **DEM editing** additional toolbar, to define operation to be performed with DEM cells.
3. [optional] To specify elevation of selected DEM cells, set the elevation value in the field to the right from the 📏 button.
4. Press and hold the **Shift** key and select area to be edited by mouse. The editing is performed automatically using parameters specified by default.


















In order to edit multiple areas, repeat the 4 step required number of times. To turn off the editing mode, selected on the 2 step, click the appropriate button again.

The system provides a combined method for selecting DEM sections. This method can be used if it is necessary to edit a DEM fragment of complex shape that describes a separate object (or a group of objects).

For this, perform the following:

1. Make sure that **Subtract from Selection** (📏) or **Invert Selection** (📏) modes are not chosen in Instruments (**Window > Toolbars > Tools**). Otherwise, choose **Normal**

- () or **Add to Selection** () (see “Object Selection” in the “[Vectorization](#)” User Manual);
2. First, select the required DEM fragment by the [building spot selection tool](#) (
 3. Press and hold **Shift** to edit the created selection using standard selection tools (, ) when **Add to Selection** () and **Subtract from Selection** () modes are enabled, *except for making the final necessary edit to the edited selection*;
 4. Choose one of the DEM editing tools (, , , , ) – see above in the current subsection);
 5. Press and hold **Shift** to make the final necessary edit to the selection (see item **3** above);

As a result, the tool for quick editing of the DEM selected in item **4** processes an area of complex shape, selected in a combined way, both using the spot selection tool () and using standard selection tools (, .

7.8.10. Elevation correction along axis line



This operation does not provide for undo. Ensure creating a [copy](#) of the DEM first.

The system provides for correcting DEM cell values along the selected linear objects.

For this, perform the following:

1. [Load](#) or [create](#) a DEM;
2. Create and select at least one linear vector object that intersects the DEM in the XY plane:

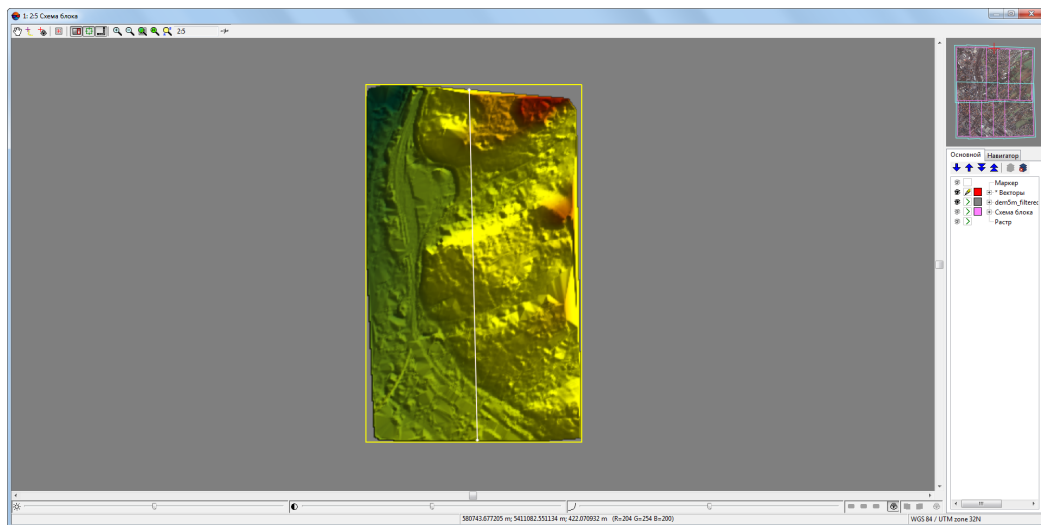


Fig. 185. A polyline intersecting a DEM in the XY plane (2D-window viewing)

3. Make the DEM layer active:

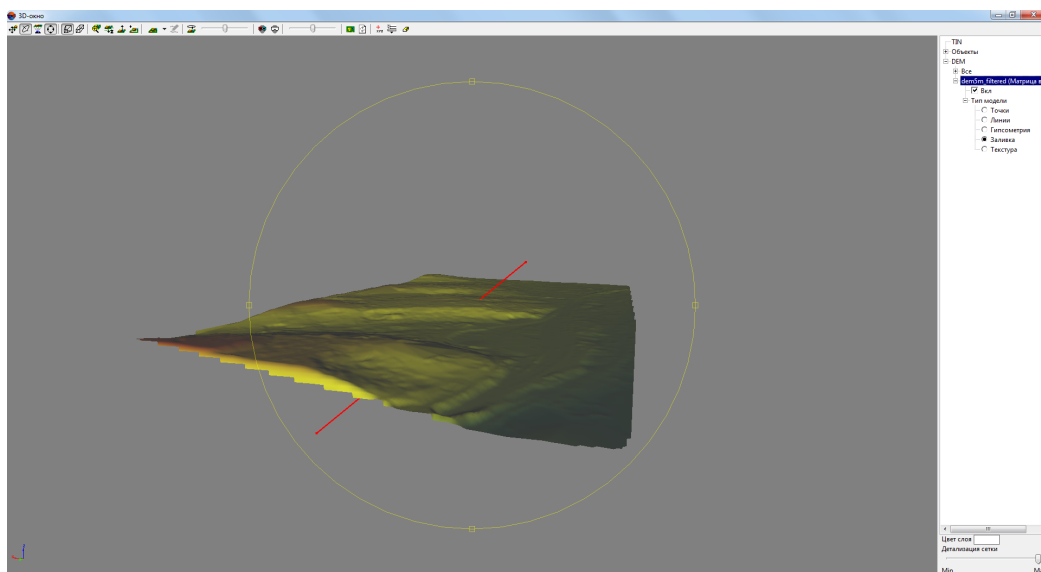


Fig. 186. A polyline intersecting DEM in XY-plane and along Z (3D-window view)

4. Choose **Terrain > DEM > Correct along vector line**. The **Axis line elevation correction** window opens:

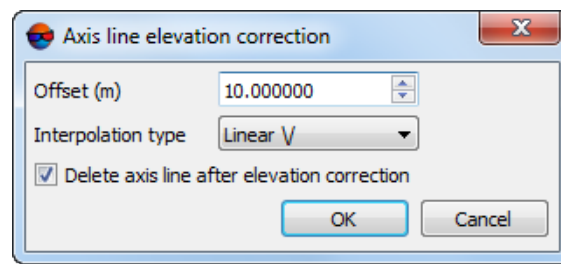


Fig. 187. The Axis line elevation correction

5. Configure the following transformation options:
 - The corridor **width** – is the size of DEM transformation area along the centerline
 - **Interpolation type**:
 - **Linear**;
 - **Squared polynomial**;
 - **Gaussian**;
 - **None**.
 - To **delete axis line after elevation correction**, set the appropriate checkbox.
6. Click OK. A warning message appears:

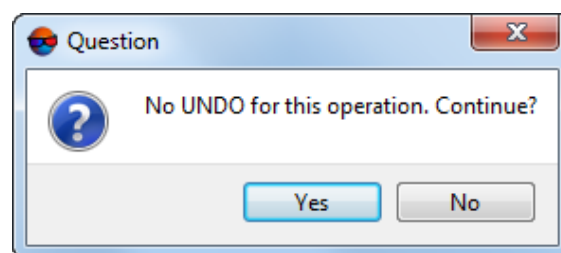


Fig. 188. An warning message

7. Click **Yes**, to confirm the operation;
8. A DEM is edited according to specified settings and used polyline/polygon vertex elevations. After successful completion of the operation, an appropriate info message is issued.:

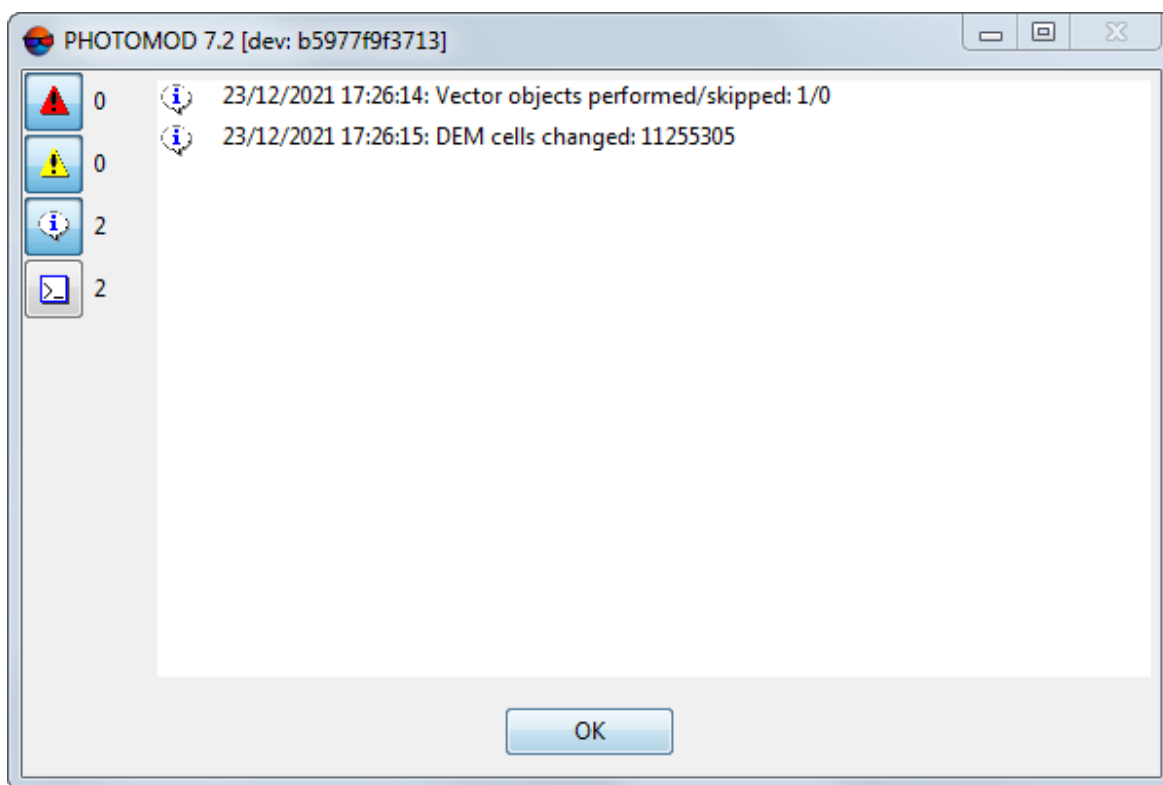


Fig. 189. An info message

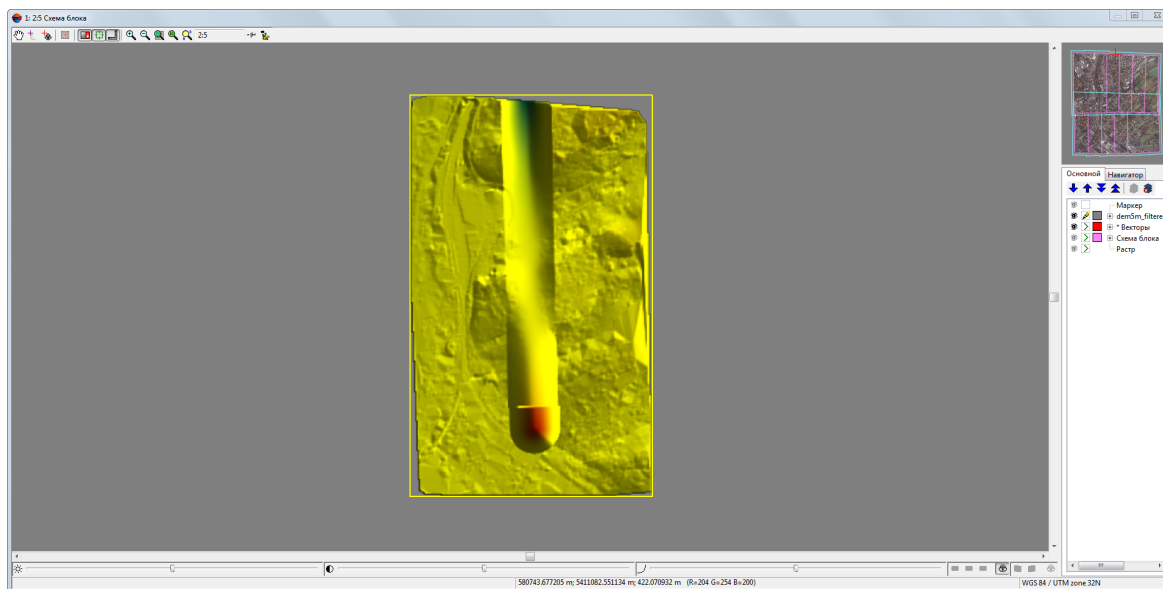


Fig. 190. Edited DEM (2D-window view)

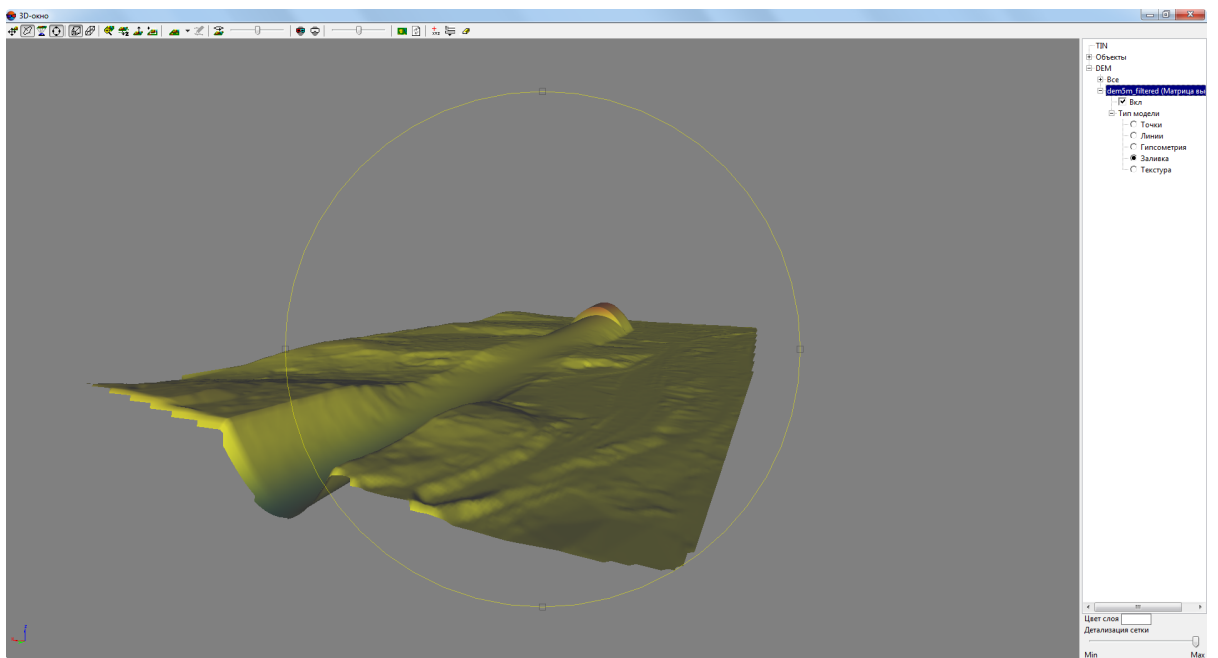


Fig. 191. Edited DEM (3D-window view). Squared polynomial interpolation. The used polyline (deleted) crossed the DEM in the XY-plane and along Z

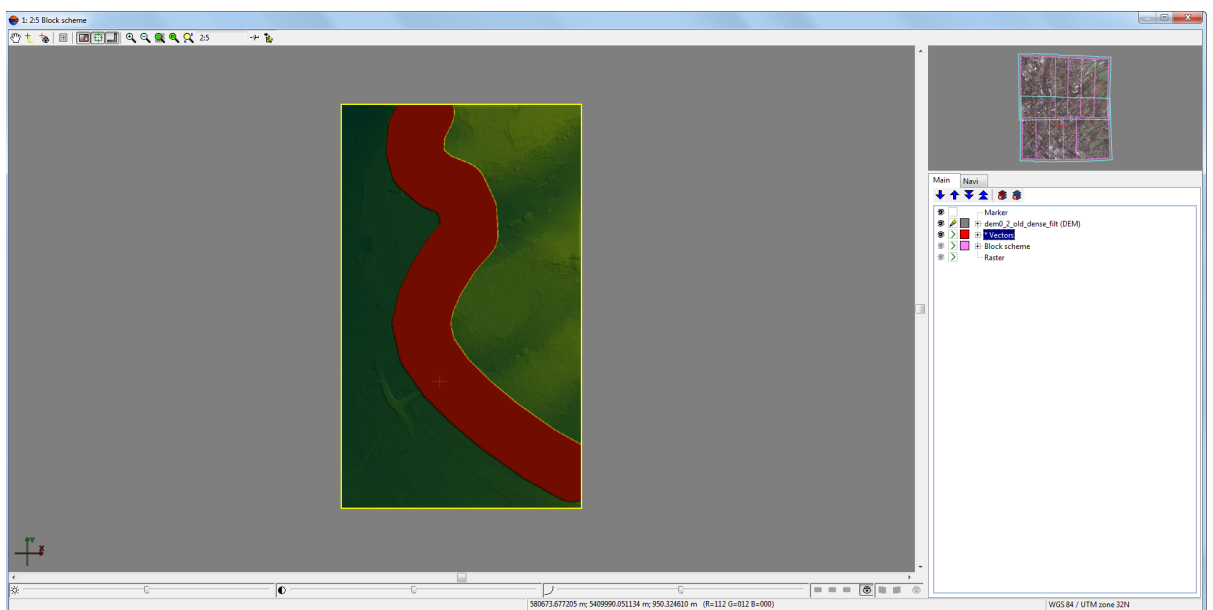


Fig. 192. Edited DEM (2D-window view)

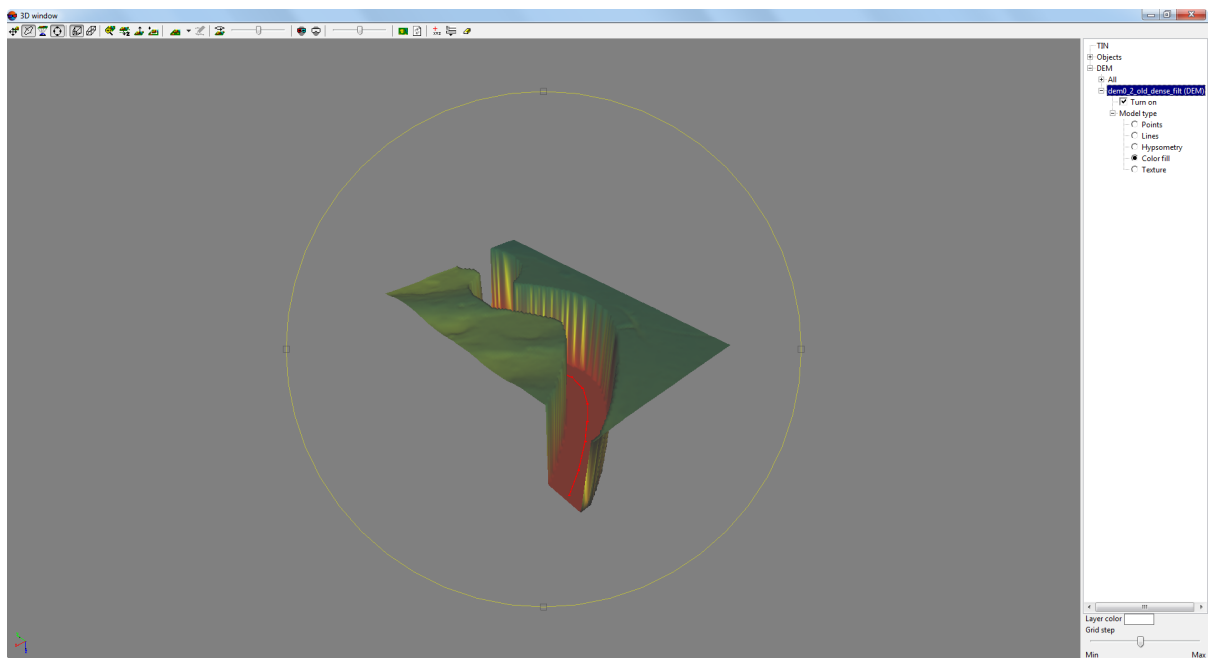


Fig. 193. DEM edited without interpolation (3D-window view). The used polyline (highlighted in red) crossed the DEM only in the XY-plane

7.9. Accuracy control of DEM creation

7.9.1. Creation of difference DEM

The system provides possibility to create difference DEM using subtraction of one DEM from the other.

To create difference DEM perform the following actions:

1. **Load** at least two DEMs to be used for subtraction;
2. [optional] Create and select vector polygon, if it is needed to create a difference DEM **within current polygon**;
3. Select **Terrain › DEM › Check accuracy › Build difference DEM**. The **DEMs comparing parameters** window opens:

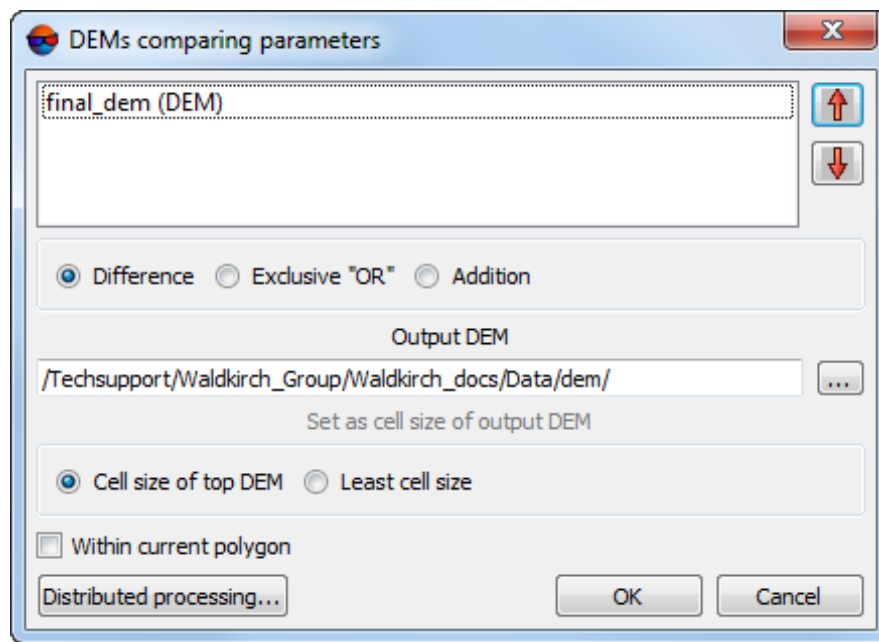





Fig. 194. Parameters of DEMs comparing

The list displays all loaded DEMs.

4. Select in the list a layer with DEM, from which the system will subtract. Use the  and  buttons to move it to the top of the list;
5. Select a way of resulting DEM creation:
 - **Difference** – the system performs subtraction of one DEM from the other. If there are null values in the one of DEMs, the output DEM will contain null values in the same cells;
 - **Exclusive 'OR'** – the output DEM cells are calculated only in areas, where one of input DEMs have a null cell values. The final sign (positive or negative) of cells elevation in output DEM depends on sequence of subtracting of one DEM from the other;
 - **Addition** – input DEMs cells values are summed up. If there are null values in the one of DEMs, the output DEM will contain null values in the same cells.
6. In the **Output DEM** section click the  button and specify name and path of output DTM in active profile resources;
7. Specify the output DEM cell size – **Cell size of top DEM** or the **Least cell size**;
8. [optional] Set the **Within current polygon** checkbox to create a difference DEM within selected polygon;

9. Select two DEMs in the list. To do this press and hold the **Ctrl** key and click the DEM list by mouse.



The DEM from which the system performs the subtraction should be higher in the list, than the subtrahend DEM.

10. Click OK. After that the system shows a message about operation successful or unsuccessful completion. In case of success the system creates new layer in the *Manager*.

To use distributed processing for difference DEM creation, 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.

For automatic positioning marker on active DEM surface the **Marker Z coordinate follows DEM in active layer** check box is on by default in the [general system's parameters](#).

7.9.2. DEMs comparing

The system provides possibility of comparing of two DEMs for the same territory to analyse their discrepancies by Z axis.

To compare DEMs perform the following actions:

1. [Load](#) DEMs to be compared;
2. [optional] Create and select a vector polygon if it is needed to compare DEMs within the selected polygon only;
3. Select **Terrain › DEM › Check accuracy › Compare DEMs**. The **DEMs compare parameters** window opens:

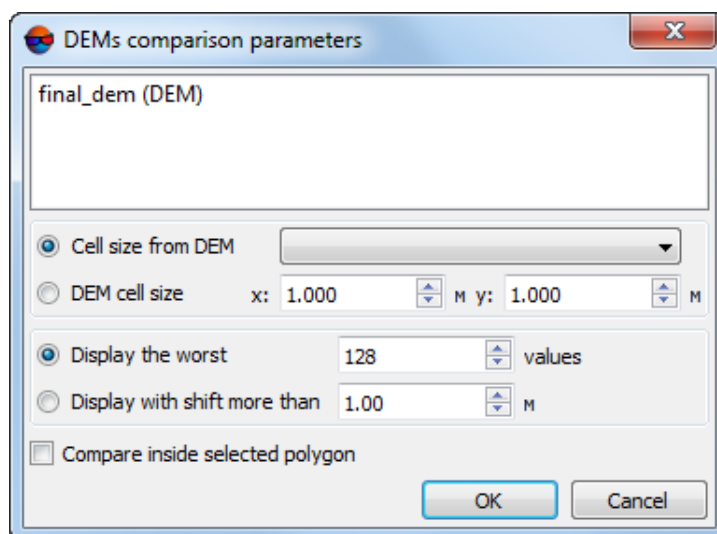


Fig. 195. Parameters of DEMs comparing

The list displays all loaded DEMs.

4. Select two DEMs in the list. To do this press and hold the **Ctrl** key and click the DEM list by mouse;
5. Set the sample step (points where elevation value comparison will take place). The systems allows to:
 - Take the **cell size from DEM** selected from the drop-down list;



Only selected DEMs are available in the drop-down list (see the section above).

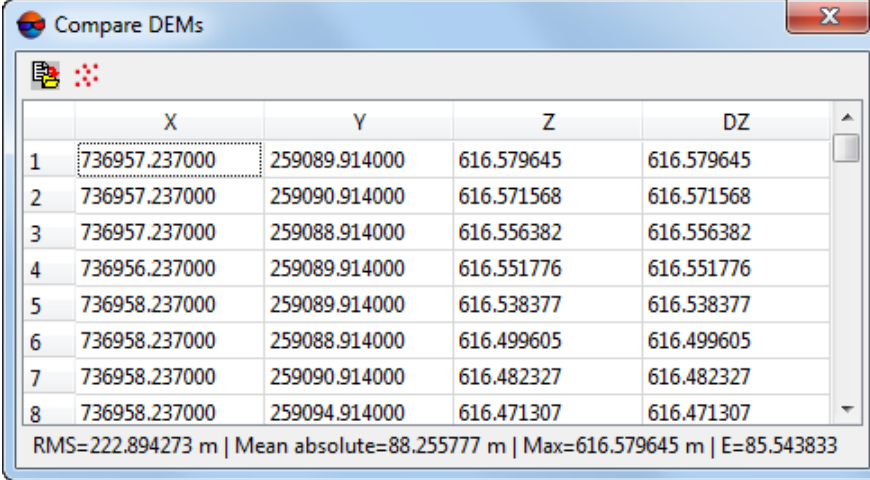
- Set the **DEM cell size** in meters along the **X** and **Y** axes.



When selecting a step, it is needed to take into account the search area size (DEM or selected polygon dimensions). Wrong step size may lead to the inability to form at least one value.

6. Set the following parameters:
 - **Display the worst..values** – allows to show a list of specified number of points with maximal error;
 - **Display with shift more than..m** – allows to show a list of points with discrepancy greater than specified.
7. Set the **Compare inside selected polygon** checkbox, if appropriate;

8. Click OK. The system starts operation of selected DEMs comparing. When the operation is completed the **Compare DEMs** window opens.



	X	Y	Z	DZ
1	736957.237000	259089.914000	616.579645	616.579645
2	736957.237000	259090.914000	616.571568	616.571568
3	736957.237000	259088.914000	616.556382	616.556382
4	736956.237000	259089.914000	616.551776	616.551776
5	736958.237000	259089.914000	616.538377	616.538377
6	736958.237000	259088.914000	616.499605	616.499605
7	736958.237000	259090.914000	616.482327	616.482327
8	736958.237000	259094.914000	616.471307	616.471307


RMS=222.894273 m | Mean absolute=88.255777 m | Max=616.579645 m | E=85.543833


Fig. 196. Parameters of DEMs comparing

The window displays a list of points coordinates, where:

- N – is point serial number;
- X, Y, Z – coordinates of erroneous point;
- DZ – discrepancy by Z.

The status line of the window displays root mean square error (RMS) of discrepancies, Mean absolute, maximal discrepancy by Z found (Max), as well as Average discrepancy among points found.

The  button allows to save the table to file with *.csv extension.

The  button allows to save the selected values of current vector layer as points. For this, select at least one value in the table. DZ values will be saved as attributes of the created points.



If there are no open vector layers, the system offers to create a new vector layer.

Double click the point record line by mouse to select the point in the DEM active layer.

7.9.3. Search for DEM areas

To perform analysis of DEM creation quality the system provides possibility of null cells search in created DEM.



It is not recommended to search for null cells when processing big data amounts.

To search for null cells in DEM perform the following actions:

1. Load or create a DEM.



If multiple DEMs are loaded into project, the search is performed in the *active* DEM layer.

2. Select **Terrain > DEM > Check accuracy > Search for DEM areas**. The **DEM areas searching parameters** window opens.

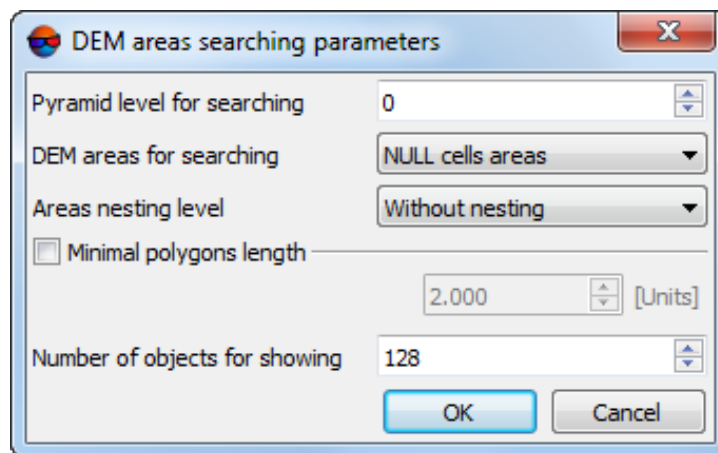


Fig. 197. DEM areas searching parameters

3. Input the image **Pyramid level** for null cells search.



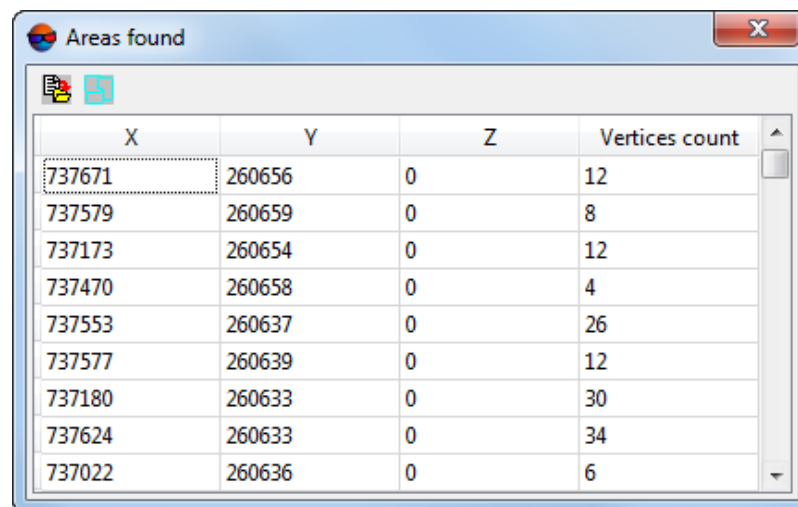
Zero pyramid level corresponds to a source image.

4. Set **DEM areas for searching**:

- **NULL cells areas** – search for DEM null cells;
- **All areas** – search for DEM null cells and cells with values;
- **Elevation areas** – search for cells with values.

5. Set **Area nesting level** to set “nested areas” search parameters (e.g. an area with null cells within an area of cells with values in its turn within a large area with null cells);


6. [optional] set the **Minimum polygon perimeter** checkbox to exclude areas having sizes lesser than the specified value from the search. Specify minimum polygon perimeter in meters;
7. Set **Number of objects for showing**, to define the number of areas to be displayed in the **Null cells** window (the preset number of areas found first will be displayed in the window).
8. Click OK. The system starts search for null cells operation. When the search is completed the system opens the **Null cells** window or displays a message about absence of null cells.




X	Y	Z	Vertices count
737671	260656	0	12
737579	260659	0	8
737173	260654	0	12
737470	260658	0	4
737553	260637	0	26
737577	260639	0	12
737180	260633	0	30
737624	260633	0	34
737022	260636	0	6

Fig. 198. Null cells list

The list of the areas with null cells (their number is limited by the **Number of objects for showing** parameter) are displayed in the window, where:

- N – is the area serial number;
- X,Y – coordinates of centers of null cells areas;
- (31) – number of vertices of the vector polygon circumscribing the area (to create a vector layer that contain points on borders of null cells area, click the  button).

The  button allows to save the list to file with *.csv extension.

To create a vector layer that contain points on borders of null cells area, click the  button.



Vector boundaries will be created for all found areas except for those excluded from search by the **Minimum polygon perimeter** option, but independent of the **Number of objects for showing** parameter.

Double click by **left mouse button** the list item in the **Null cells** window moves the marker to the appropriate area found during the search.

To edit DEM with null cells the system provides the following features:

- various methods of null cells filling (see [Section 7.7](#));
- alignment of selected DEM fragment by elevation (see [Section 7.7](#));
- vector objects editing and DEM rebuilding, when base vector layers are used for DEM creation.

7.9.4. DEM control by TIN

The system provides possibility to control created DEM by TIN. The control is performed by calculation of source TIN nodes discrepancies from created DEM.

To control created DEM perform the following actions:

1. Select **Terrain › DEM › Check accuracy › Check against TIN**. The **Parameters of check DEM using TIN** window opens.

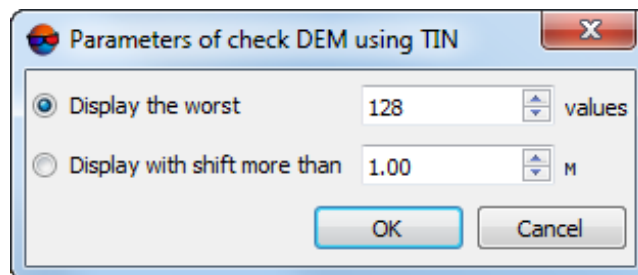
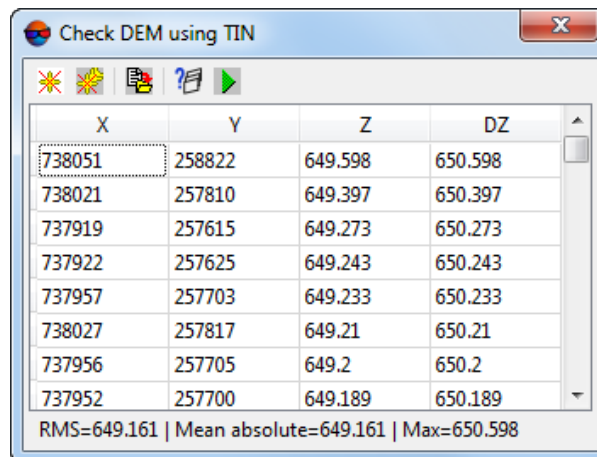


Fig. 199. Parameters of DEM accuracy control by TIN

2. Set the following parameters:
 - **Display the worst..values** – allows to show a list of specified number of TIN nodes with maximal error;
 - **Display with shift more than..m** – allows to show a list of points with discrepancy from DEM by Z greater than specified.
3. Click OK. The system starts operation of selected DEMs comparing. When the operation is completed the system opens the **Check DEM using TIN** window.



X	Y	Z	DZ
738051	258822	649.598	650.598
738021	257810	649.397	650.397
737919	257615	649.273	650.273
737922	257625	649.243	650.243
737957	257703	649.233	650.233
738027	257817	649.21	650.21
737956	257705	649.2	650.2
737952	257700	649.189	650.189

RMS=649.161 | Mean absolute=649.161 | Max=650.598

Fig. 200. Results of DEM control by TIN

The window displays a list of points coordinates, where:

- N – is serial number;
- X, Y, Z – coordinates of erroneous point;
- DZ – discrepancy by Z between DEM and TIN.






The status line of the window displays root mean square error (RMS) of discrepancies, Mean absolute, maximal discrepancy by Z found (Max) on points found.

The  button allows to save the list to file with csv extension.

Double click the point record line by mouse to select the point in the DEM active layer.



A list of erroneous nodes includes those located on all stereopairs of current project.

- [optional] To change the list display parameters, click the  button. In order to display changes in the list click the  button.
- To edit TIN using the list the system provides the following features:
 - to remove selected TIN node, click the  button or press the **Delete** key;
 - to remove all nodes, where the error is more or equal to the error selected in the list, click the  button.
- Click the  button, to re-build the DEM considering all changes.

7.9.5. DEM control by vector objects

The system provides possibility to control DEM creation quality by vector objects. During comparing vector objects with DEM the system compares Z-coordinates of points or vector objects vertices with elevation in corresponding points of the DEM. This vector objects layer should not be used as a base layer during this DEM or TIN creation.

DEM control by vector objects is needed to estimate accuracy of imported DEM or, vice versa, of vector objects by 'master' DEM.

To control DEM creation against vector objects perform the following actions:

1. **Load** DEM;
2. Load vector layer;
3. Select **Terrain > DEM > Check accuracy > Check against vector objects**. The **Parameters of check DEM by vectors** window opens.

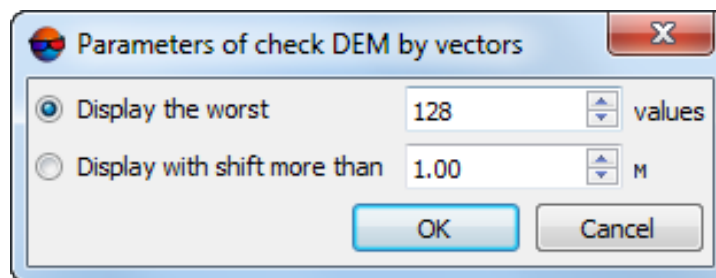
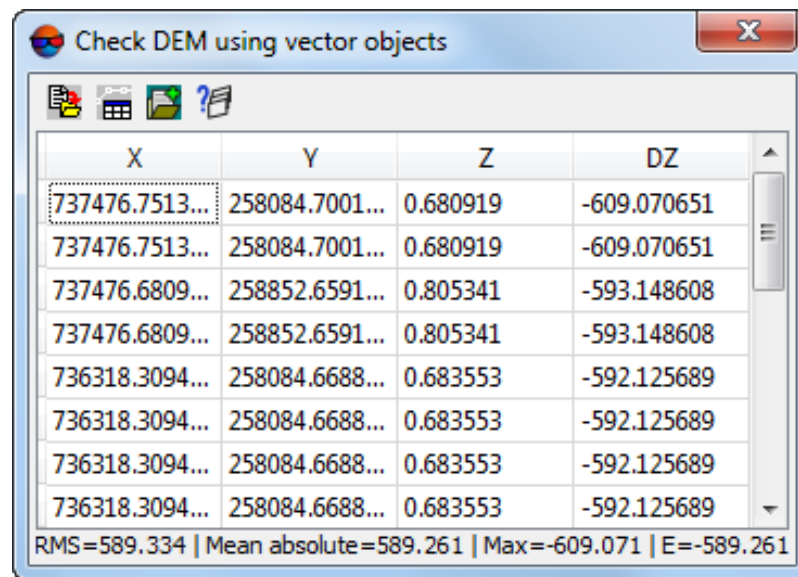


Fig. 201. Parameters of accuracy control by vectors

4. Set the following parameters:
 - **Display the worst..values** – allows to show a list of specified number of points with maximal error;
 - **Display with shift more than..m** – allows to show a list of points with discrepancy from DEM by Z greater than specified.
5. Click OK. The system starts operation of selected DEMs comparing. When the operation is completed the system opens the **Check DEM using vector objects** window.



X	Y	Z	DZ
737476.7513...	258084.7001...	0.680919	-609.070651
737476.7513...	258084.7001...	0.680919	-609.070651
737476.6809...	258852.6591...	0.805341	-593.148608
737476.6809...	258852.6591...	0.805341	-593.148608
736318.3094...	258084.6688...	0.683553	-592.125689
736318.3094...	258084.6688...	0.683553	-592.125689
736318.3094...	258084.6688...	0.683553	-592.125689
736318.3094...	258084.6688...	0.683553	-592.125689

RMS=589.334 | Mean absolute=589.261 | Max=-609.071 | E=-589.261

Fig. 202. Results of DEM control by vector objects

The window displays a list of points coordinates, where:

- N – is serial number;
- X, Y, Z – coordinates of erroneous point;
- DZ – discrepancy by Z between DEM and vector objects vertices.


The status line of the window displays root mean square error (RMS) of discrepancies, Mean absolute, maximal discrepancy by Z found (Max), as well as Average discrepancy among points found.


Double click the point record line by mouse to select the point in the DEM active layer.



A list of erroneous nodes includes those located on all stereopairs of current project.


The  button allows to save the list to file with *.csv extension.

The  button allows to select those vector objects within the layer, the vertices of which were selected in the table.

The  button allows to save max DZ values for each vector object in the vector object attributes.



The name of the attribute created in the process of saving is MAX_DZ.

The  button allows to specify the **attribute selection settings** to include attributes into a table of the results of DEM accuracy control:

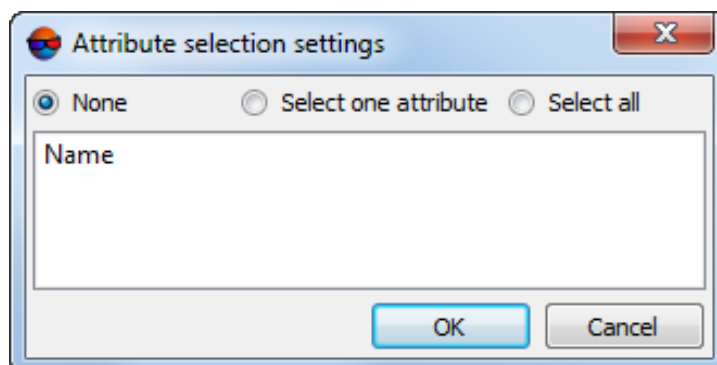


Fig. 203. The Attribute selection settings window

Using this window, user can include into the table any vector object attributes used for the accuracy control, if these attributes were previously created. The window allows to **select one attribute** to be included, **select all**, or **none**.

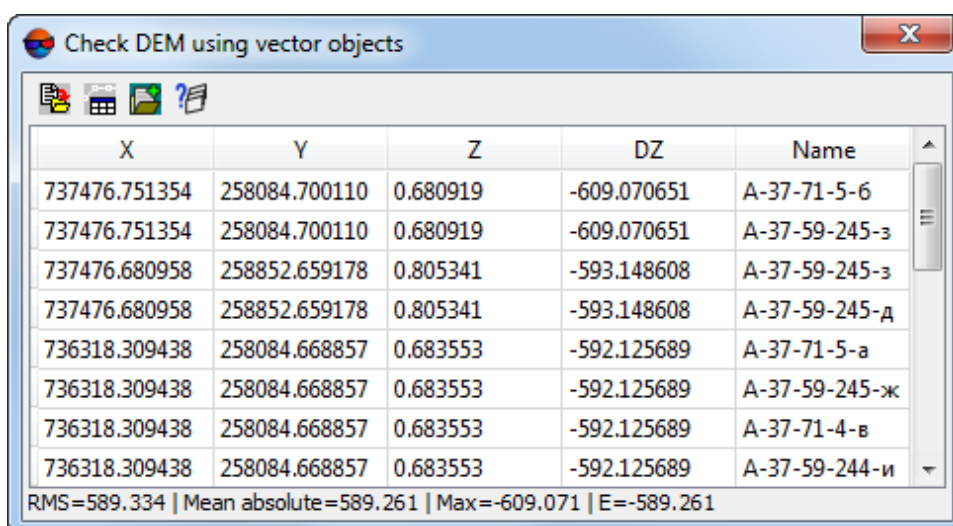



Fig. 204. Including previously created attributes into the table

For example, to include the MAX_DZ attribute value into the table of accuracy control results, perform the following:

1. Create the attribute ();
2. Repeat the accuracy control operation (**Terrain > DEM > Check accuracy > Check against vector objects**);
3. Include this attribute into the table using the **attribute selection settings** window.

7.9.6. DEM control by triangulation points

The system provides possibility to control DEM creation quality by triangulation points. During comparing triangulation points layer with DEM the system compares Z-coordinates of triangulation points with elevation in corresponding points of the DEM. This triangulation points layer should not be used during this DEM creation.

To do this use the **Terrain > DEM > Check accuracy > Check against TIN** menu item.



Triangulation points must have the adjusted coordinates (**X, Y, Z**).



It is not necessary to create points vector layer.

The **Accuracy control** window includes a table of all points (GCP/check/tie), acquired on on aerial triangulation step (see the “[Aerial triangulation](#)” User Manual).

	Name	Type	Use	X	Y	Z	Ez
1	163	GCP	<input checked="" type="checkbox"/>	737988.803	259849.352	586.238	-0.396
2	40169P	GCP	<input checked="" type="checkbox"/>	736405.463	258080.383	589.470	-0.332
3	6719	GCP	<input checked="" type="checkbox"/>	737810.890	257589.754	633.267	1111744.267
4	6720	GCP	<input checked="" type="checkbox"/>	736702.692	258651.487	586.604	-0.180
5	779	GCP	<input checked="" type="checkbox"/>	737124.870	257793.550	612.432	-0.250
6	970	GCP	<input checked="" type="checkbox"/>	736273.121	259010.315	581.185	-0.058
7	~1	Tie	<input checked="" type="checkbox"/>	737216.877	258832.574	589.083	-0.014
8	~100	Tie	<input checked="" type="checkbox"/>	736020.444	258043.728	600.153	-0.497
9	~10013	Tie	<input checked="" type="checkbox"/>	737061.169	258985.709	584.357	-0.074
10	~10013 [1]	Tie	<input checked="" type="checkbox"/>	737061.188	258985.715	584.341	-0.090
11	~10013 [2]	Tie	<input checked="" type="checkbox"/>	737061.188	258985.715	584.341	-0.090
12	~10013 [3]	Tie	<input checked="" type="checkbox"/>	737061.188	258985.715	584.341	-0.090

RMS: 4.468 Greatest: -76.197

Fig. 205. Accuracy control using triangulation points

Checkboxes in the toolbar at the top of the window are used to customize the display of triangulation points in the list according to their types and properties (see the “Point properties” chapter in “[Aerial triangulation](#)” User Manual):

- **Gr.control** – set the appropriate checkbox to display points of this type in the table;;
- **Check** – set the appropriate checkbox to display points of this type in the table;
- **Tie** – set the appropriate checkbox to display points of this type in the table;

- **Targeted** – set the appropriate checkbox to display **targeted** points in the table.



Only those types of triangulation points that present in this project are available for selection.

To use the **adjusted** points to check the accuracy – set the appropriate checkbox.




Clear this checkbox to perform accuracy control by triangulation points disregarding adjustment.


To display in the table only the points **inside DEM**, set the appropriate checkbox.

The **Used points** field is used to display number of points used for calculation of DEM deviation by Z-coordinate (in the **Use** column the ☒ is set).

The table of triangulation points contains the following columns:

- Name – point number;
- Type – point type: GCP, check or tie point;
- Use – data about point use for calculating of DEM deviation by Z-coordinate:
 - ☒ – the triangulation point was used;
 - ☐ – the triangulation point was not used;
 -  – the triangulation point cannot be used, since it was not used for adjustment.
- X, Y, Z – adjusted coordinates of triangulation points;
- Ez – value of DEM deviation from the triangulation point by Z-coordinate.

Status panel is used to display values of root mean square error (RMS) and a value of maximal error of DEM deviation by Z-coordinate (Max).

The  button allows to display in the list a point with maximal error of deviation from triangulation points by Z-coordinate.

The  button allows to display detailed report on accuracy control.

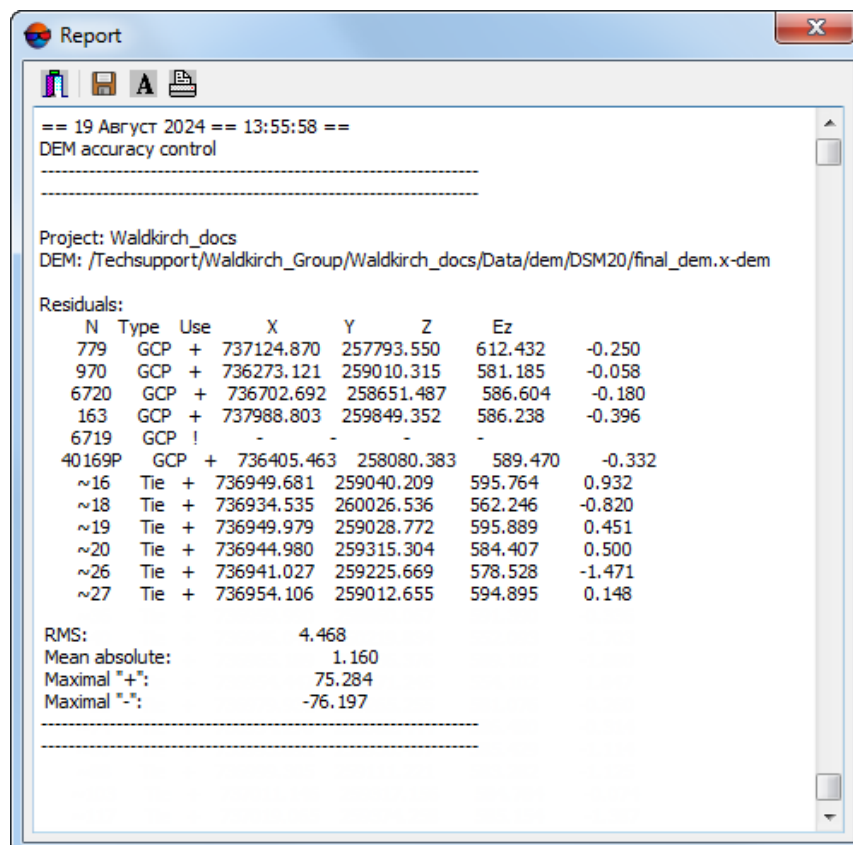


Fig. 206. Report on accuracy control using triangulation points

The top panel of the **Report** window contains the following buttons:

- – allows to close report window;
- – allows to save a report to file system;
- – allows to change font, style, size, and color of report text;
- – allows to print a report.

7.9.7. Creation of histogram by DEM

The system allows to create histogram by DEM.



A histogram can be, for example, a convenient tool to analyze a [difference DEM](#).

To do this use the **Terrain > DEM > Check accuracy > Create histogram by DEM** menu item. The **Histogram** window opens.

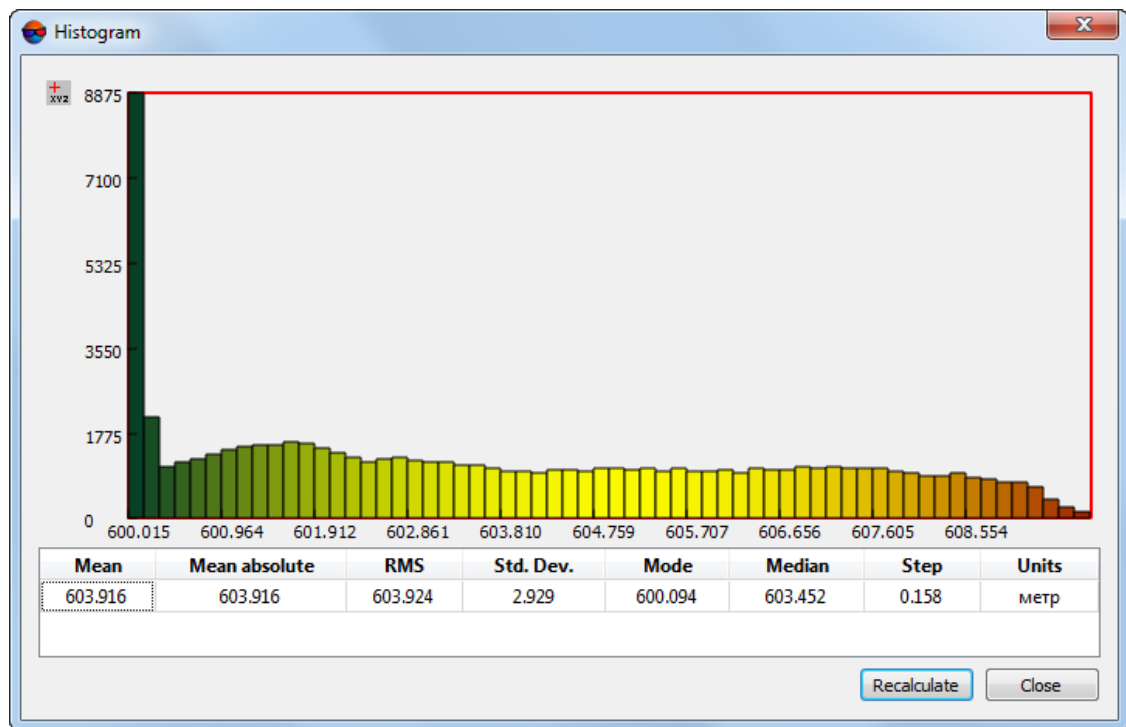


Fig. 207. Histogram

A histogram displays the dispersion of the number of cells (Y-axis) within the working elevation range (X-axis).



The histogram palette is determined by the user-defined DEM palette (see the “Settings of DEM display” chapter in “[General system's parameters](#)” User Manual).

There is a table at the bottom of the **Histogram** window that contain the following information:



When building a histogram for the first time the **work elevation range** is equal to the DEM elevation range by default (i.e. the histogram to be created for all DEM cells). Later (see below) the **work elevation range** can be narrowed, e.g., to exclude evidently false values (so called peaks) from the data used for histogram building.

- The arithmetical **mean** elevation of data used for histogram building is the sum of cell elevations divided by the number of cells. It depends on the **work elevation range** that is used when building a histogram;
- The **mean absolute** is the sum of absolute values of cell elevations divided by the number of cells. It depends on the **work elevation range** that is used when building a histogram;
- The **RMS** (root-mean-square deviation) is the square root of the sum of the squares of the cell elevations divided by the number of cells. It depends on the **work elevation range** that is used when building a histogram;

- The **std. dev.** is the square root of the sum of the squares of deviations divided by the number of cells. It depends on the **work elevation range** that is used when building a histogram;



A *deviation* is the difference between the cell elevation and the arithmetic mean elevation in the data used (see the first item above).

- The **mode** is a value of elevation most frequently repeated among the data used for histogram building. It depends on both parameters of histogram building **work elevation range** and **step**);
- The **median** divides a series of ordered elevation values in two, with an equal number of values both above and below the median (i.e. to the left and to the right of the X-axis). It depends on both parameters of histogram building (**work elevation range** and **step**);
- The **step** is calculated automatically or set manually;
- Project **units**.

To **recalculate** the histogram with the custom **Work elevations range** and histogram **step** – click the appropriate button. The **Histogram parameters** window opens.

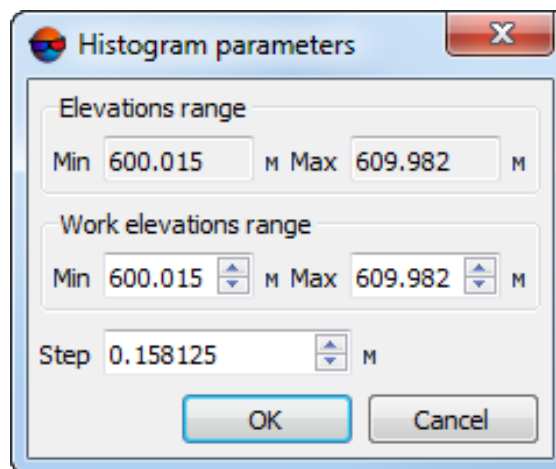



Fig. 208. Accuracy control using triangulation points


The top of the window is used to display DEM **Elevations range**. The bottom part of the window is used to specify histogram creation parameters. To create histogram set the **Work elevations range** and histogram **step** and click OK. The **Histogram** window opens.



A **step** is to be selected taking into account the specified **work elevations range**. A very large step value will be automatically reduced by the system to the maximum allowable value relative to the current work elevations range.

To display a pop-up help containing the exact number of cells (**Y**-axis) and boundaries of the elevation working range segment (**X**-axis) when pointing the marker at a certain column of a diagram, click .



To disable the pop-up help mode, click the  button again

7.9.8. Creating statistics file by DEM

The system allows to get statistics data by DEM. Statistics data is provided as a file in *.csv format and includes information on distribution of DEM cells in specified range of heights, as well as information on ratio of specified cells number to the all DEM cells number.

To create a statistics file, perform the following:

1. Create or load DEM;
2. Select **Terrain > DEM > Check accuracy > Get statistics by DEM**. The **Get statistics by DEM** window opens;

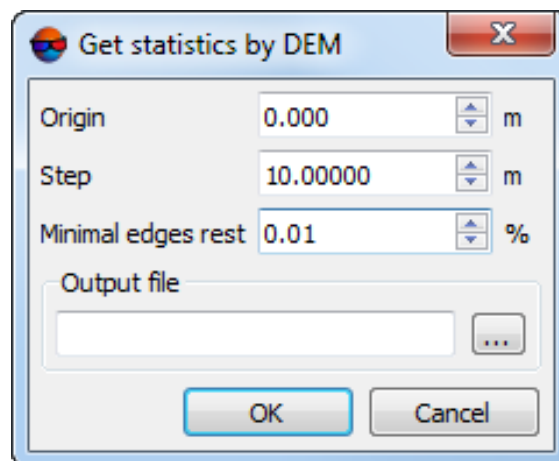



Fig. 209. Window of getting statistics by DEM

3. Click  button to specify a location of the statistics file;
4. Set the following parameters:
 - **Origin** – a height in meters started from which a statistic on distribution of DEM cells number will be collected;
 - **Step** – a size of height ranges in meters, for which a statistic on distribution of DEM cells number will be collected;

- **Minimal edges rest** – permissible deviation between the 1 and the sum of the **Percent** parameter values in statistic file (see below). The **Percent** parameter is the ratio of the cell number in each of the ranges to the total number of DEM cells.

5. Click **OK**. Statistic file in *.csv format is created in the specified folder.



For example, when the following parameters are specified:

- **Origin** – 0.0 m.
- **Step** – 10.0 m.
- **Minimal edges rest** – 0.01.

a file of statistics by DEM looks like this:

Range,Number of points,Percent

< 0,0,0

0 - 10,0,0

10 - 20,0,0

<...>

550 - 560,0,0

560 - 570,19402,0.783919

> 570,5348,0.216081

where:

- **Range** – is range of heights for which a statistic was collected, in meters.
- **Number of points** – is a number of cells in the corresponding range.
- **Percent** – is a ratio of specified number of cells to the all DEM cells number.

respectively, in the example above all the DEM cells are placed in two height ranges only:

- 560 – 570 meters: 19402 cells, which are about ~ 78% of the total;
- 570 – 580 meters: 5348 cells, which are about ~ 22% of the total.



Thus, a total of **Percent** parameter values must be 1, where a specified **Minimal edges rest** parameter is an acceptable value of deviation from this number.

7.9.9. Lists of DEM min/max values

The system provides possibility to obtain DEM statistical data organized as the lists of minimum and maximum values of DEM.

To do this, perform the following actions:

1. Create or download DEM;
2. Choose **Terrain › DEM › Check accuracy › Show DEM min/max elevations**. The **Number of elevations for showing** window opens;

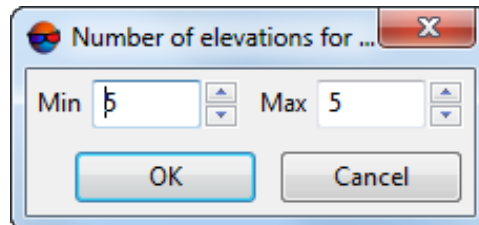


Fig. 210. The Number of elevations for showing window

3. In the **Min** field, specify the number of minimum DEM values displayed (from 1 to 100);
4. In the **Max** field, specify the number of maximum DEM values displayed (from 1 to 100);
5. Click **OK**. The **DEM min/max elevations** window opens. The window contains lists of minimum and maximum DEM values;

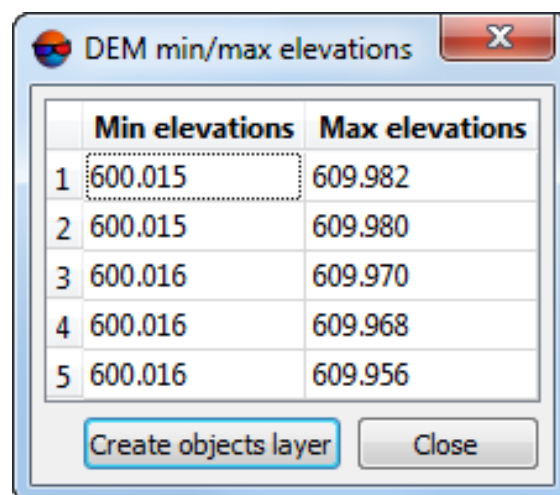


Fig. 211. The DEM min/max elevations window

6. [optional] click **Create objects layer** button to create a layer with points which mark the locations of DEM cells with minimum and maximum values from the obtained lists;

Point names are as follows: 529.638(min) and 671.232(max), where 529.638 and 671.232 are the elevations, (min) or (max) is the membership of an elevation in minimum or maximum elevation list, respectively.

7. To close the **DEM min/max elevations** window, click **Close** button.

7.9.10. Subtracting the matrix values from the constant height

The system allows to subtract the matrix values from the constant height (zero by default).

For this, perform the following:

1. Create or load a DEM;
2. Choose **Terrain › DEM › Check accuracy › Subtract the matrix values from the constant height**. The **Subtract the matrix values from the constant height** window opens:

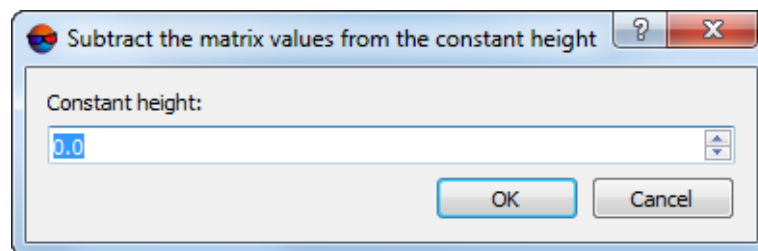


Fig. 212. The Subtract the matrix values from the constant height window

3. [optional] In the **Constant height** input field enter the appropriate value to subtract the matrix values from the nonzero constant height;
4. Click OK. By default, the output DEM is saved in the same folder of active profile resources as the input DEM. It has the same name with the `_invert_level` suffix, i.e. it has a name of the following kind: `<Input DEM name_invert_level.x-dem`.

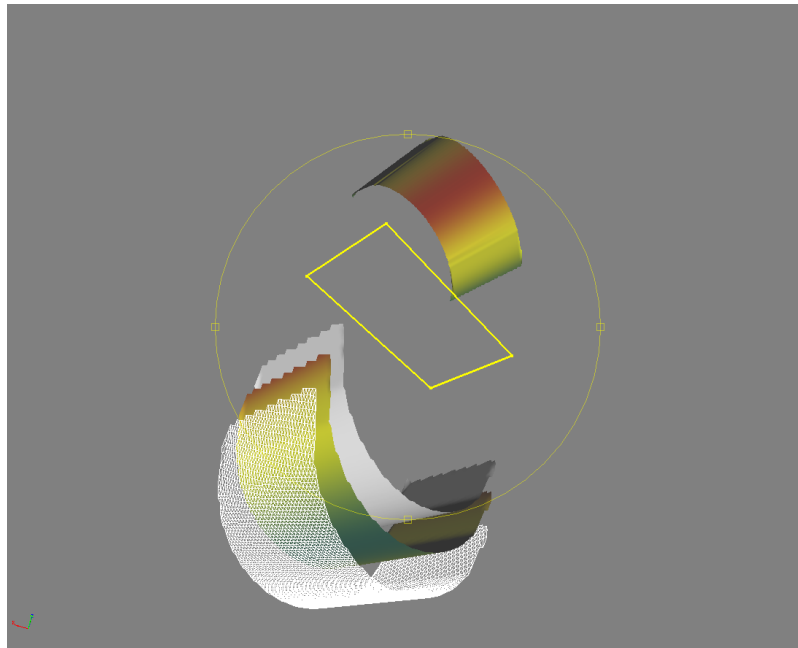


Fig. 213. Input DEM (top), surface with constant elevation (denoted by a rectangular polygon), output DEMs, created with different **Constant height** values (bottom)

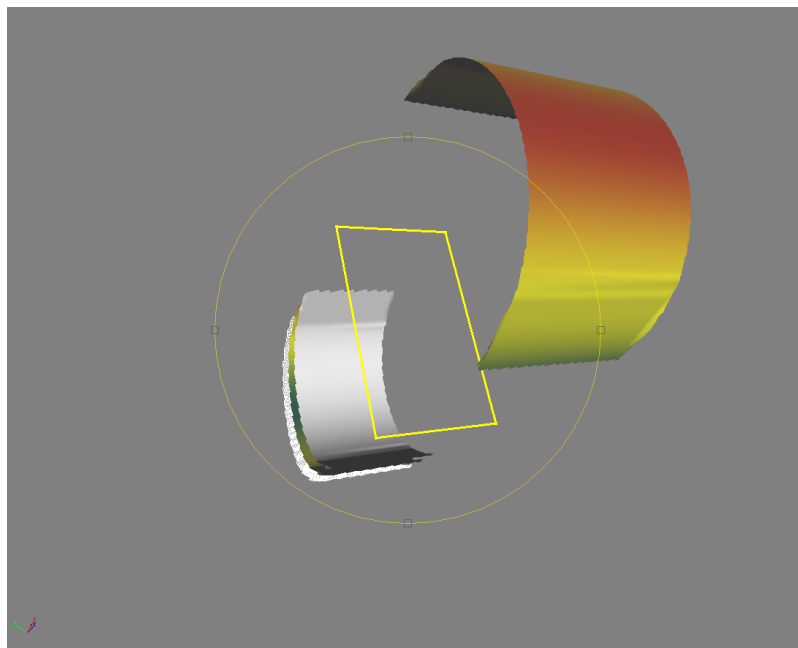


Fig. 214. Input DEM (top), surface with constant elevation (denoted by a rectangular polygon), output DEMs, created with different **Constant height** values (bottom)

7.9.11. Slopes map creation

The system provides possibility to create a slopes map, which represents the rate of change of elevation for each DEM cell.

Slope is the rate of maximum change in Z-value from each cell of the input DEM.

For each cell, the system calculates the maximum rate of change in value from that cell to its neighbors.

The maximum change in elevation over the distance between the cell and its eight neighbors identifies the steepest downhill descent from the cell.

The lower the slope value, the flatter the terrain; the higher the slope value, the steeper the terrain.

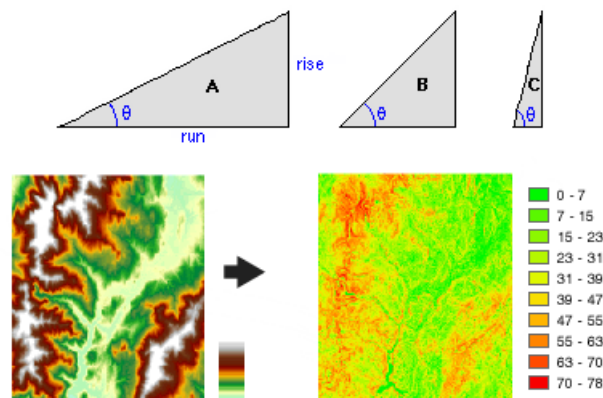


Fig. 215. Input DEM (left) and output slope map (right). Steeper slopes are shaded red on the output slope map. θ – slope (in degrees).

During the slope map creation the slope value of the center cell of the moving window shown below will be calculated.

The rates of change (delta) of the surface in the horizontal (dz/dx) and vertical (dz/dy) directions from the center cell determine the slope. The values of the center cell and its eight neighbors determine the horizontal and vertical deltas.

The neighbors are identified as letters from a to i, with e representing the cell for which the aspect is being calculated.

The basic algorithm used to calculate the slope is:

Slope (in degrees) = $\text{ATAN} (\sqrt{[dz/dx]^2 + [dz/dy]^2}) \cdot 57.29578$, where:

- $[dz/dx] = ((c + 2f + i) - (a + 2d + g)) \div (8 \cdot \text{<cell_size_by_x>});$
- $[dz/dy] = ((g + 2h + i) - (a + 2b + c)) \div (8 \cdot \text{<cell_size_by_y>});$
- $57.29578 = 180 \div \pi.$

a	b	c	50	45	50	59	56	59
d	e	f	30	30	30	71	75	70
g	h	i	8	10	10	60	63	57

Fig. 216. Surface scanning window (left). Slope example input (center). Slope example output (right).

If there is a DEM cell in the neighborhood with an empty value, the Z-value of the center cell will be assigned to this cell. At the edge of the DEM at least three cells (outside the DEM's border) will be empty. These cells will be assigned the center cell's Z-value. The result is a flattening of the 3×3 plane fitted to these edge cells, which usually leads to a reduction in the slope near the DEM's borders.

To create a slopes map perform the following actions:

1. **Load** at least one DEMs;
2. Select **Terrain › DEM › Calculate slope map**. The **Prepare slope map** window opens:

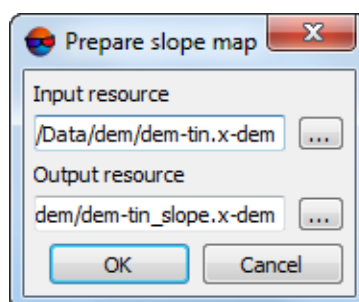


Fig. 217. Parameters slope map calculation

3. In the **Input resource** section click the **...** button to select input DEM in active profile resources.
4. Input path to save file to the **Output resource** field or click the **...** button, to select name and path in active profile resources to save output slope map.



By default, the slope map (as a DEM) is saved in the same active profile resource folder as the input DEM. It has the same name and the `_slope` suffix, i.e. has the following name: `<input DEM name>_slope.x-dem`.

5. Click OK. The system creates new layer in the *Manager*.

7.9.12. Building DEM elevation profile along a linear object

The system allows to build a DEM elevation profile along a linear object.

For this, perform the following:

1. [Load](#) or [create](#) DEMs;
2. Create and select **one** linear object that crosses DEMs in the XY-plane;
3. Choose **Vectors › Geometry › Vertical DEM profile along selected line**.

The window opens that contains an elevation profile of all DEMs along the selected linear object which crosses them. DEM elevations in meters are at the Z-axis, and polyline horizontal scan in the project units is at the L-axis. Elevation profiles of each DEM are marked with different colors. In the lowest part of the elevation profile window there is a table with the list of DEMs, containing the following columns:

- DEM – contains information about layer name and layer type;
- A separate Color corresponding to the each DEM;



The number of colors in the palette is 12. Building elevation profiles for more than 12 DEMs at once will result in doubling colors in the Elevation profile window.



To assign the DEM profile line a customized color, double click the object's color in the line Color column and select the color from the standard palette.

- Z – current DEM heights;
- dZ – the current height differences between the base DEM from the list and other DEMs;
- Base – To set a selected DEM as base, click the necessary line in the table by the right mouse button and select the appropriate item in the dropdown menu.



When moving the cursor along the horizontal polyline scan in the **Elevation profile** window, the current **L** value is displayed in the vicinity of the cursor, and current **Z** and **dZ** values in the table at the bottom of the **Elevation profile** window.



Fig. 218. The window of DEM elevation profile along a linear object



Table 9. The elevation profile window functionalities

Buttons	Function
	zoom in
	zoom out
	gets to 1:1 scale
	change scale along the L -axis, using Mouse wheel
	change scale along the Z -axis, using Mouse wheel
	to save the window content as an image with *.tif, *.bmp, *.png or *.jpg extensions in the <i>Windows</i> file system
	to enable a mode in which cursor movements along the horizontal scan of the polyline in the elevation profile window are synchronized with marker movements along the polyline in the 2D window in the <i>fixed marker mode</i> (which turns on automatically)

The system also allows to zoom in/out the elevation profile window content rotating the mouse wheel.

See the description of the *fixed marker mode* in the “[General information](#)” User Manual.

The system also allows to set the profile’s visibility:

-  – profile is visible in window;
-  – profile is invisible.

7.9.13. Assigning extreme DEM heights within the polygon as attributes

The system provides for analyzing DEM heights within polygons. During this operation, each of the polygons that limit a fragment of the studied DEM is assigned four extra attributes:

- DEM_Zmin – minimum DEM height within the polygon;
- DEM_Zmax – maximum DEM height within the polygon;
- DEM_Zmed – median DEM height within the polygon;
- DEM_Zmean – average height within the polygon.

To assign heights as polygon attributes, perform the following:

1. Load the vector layer and DEM into the project;



To ensure this function operates correctly, make sure that only one vector layer and one DEM are open in the project at the same time.

2. [optional] Select the desired polygons;
3. Choose **Vectors** › **Attributes** › **Assign height by DEM**. The **Write attributes** window opens:

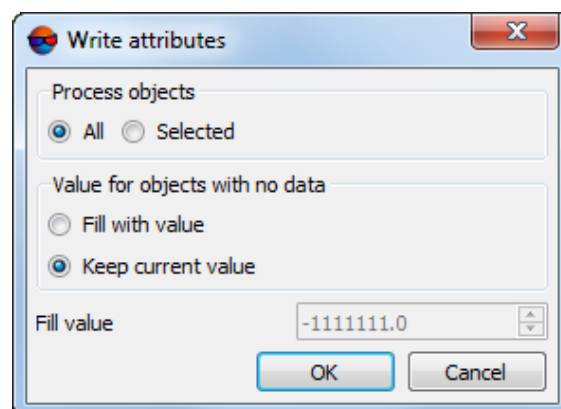


Fig. 219. Options for assigning object heights as attributes

4. Choose **processed objects** – **all** or **selected**;
5. Choose the **value for objects with no data**:

- **Fill with value** (specified by user) – the **fill value** will be assigned as DEM_Zmin, DEM_Zmax, DEM_Zmed and DEM_Zmean attributes;
 - Enter the **fill value**.
 - **Keep current value** – vector objects will remain the same.
6. Click OK. As a result, the heights DEM_Zmin, DEM_Zmax, DEM_Zmed and DEM_Zmean are assigned as attributes to all or selected objects within the vector layer.

7.10. DEM saving

The system allows to save a DEM to file with *.x-dem extension. File name and target folder to place DEM in active profile resources should be specified prior to the DEM creation.

To save opened DEM to a new file the system provides the **Terrain › DEM › Save copy...** menu item.

To save DEM area select a part of the DEM (see the chapter “Objects selection modes” in the “[General information](#)” User Manual) and choose **Terrain › DEM › Save selection**.

To save a DEM as a raster file with geodetic reference perform the following actions:

1. [Load DEM](#).
2. Select **Terrain › DEM › Save as georeferenced raster...** The **Settings** window opens.

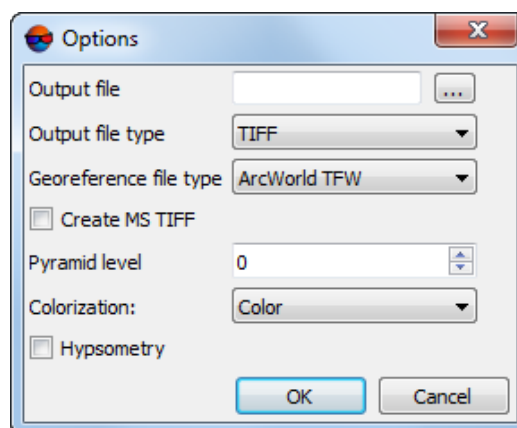


Fig. 220. Parameters of saving DEM to raster file

3. In the **Output file** section click the **...** button and specify name and target folder of output DEM out active profile resources.

4. Select **Output file type** – TIFF or BMP.
5. Select **Georeferenced file type**:
 - **No** – if there is no georeferenced files;
 - **PHOTOMOD GEO**;
 - **ArcWorld TFW**;
 - **MapInfo TAB**.
6. [optional] To create additional interior system file in MS TIFF format, that allows to use output raster file in the system without any conversion, set the **Create MS TIFF** checkbox on.
7. [optional] To save as an image the level of the pyramid that is different from the original, specify the **Layer number**.
8. Select the **Colorization** of output file – **Color** or **Grayscale**.
9. [optional] To increase the detail of the display when save the DEM as a raster file, set the **Hypsometry** checkbox on.
10. Click OK. After that the system creates file with raster representation of the DEM, and georeferenced file as well.

7.11. DEM recovering

The system provides possibility to recover DEM's consistency, which may be disrupted by applying of DEM editing or filtering operations.

To recover a DEM perform the following actions:

1. [Load a DEM](#) in the project.
2. Select **Terrain › DEM › Recover**. The **Rebuild DEM pyramid** window opens.

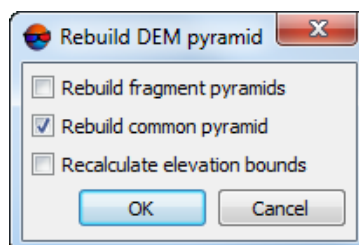


Fig. 221. DEM recovering

3. Select the following parameters of DEM recovering:


- **Rebuild fragment pyramids** – allows to rebuild a DEM by fragments (tiles) if the DEM consist of them;
- **Rebuild common pyramid** (is set on by default) – allows to rebuild common pyramid for DEM;
- **Recalculate elevation bounds** – allows to change DEM colour representation after changing its boundaries elevation.

4. Click OK. After that the system rebuilds DEM and shows a message about successful or unsuccessful DEM update.

7.12. DEM loading

The system allows to load saved DEM both from active profile resources and from file that is out of active profile resources.

In order to load a DEM from active profile resources, do the following:

1. Select **Terrain > DEM| Load (Ctrl+O, D)** or click the  buttons of the main toolbar. The **Open** window opens.

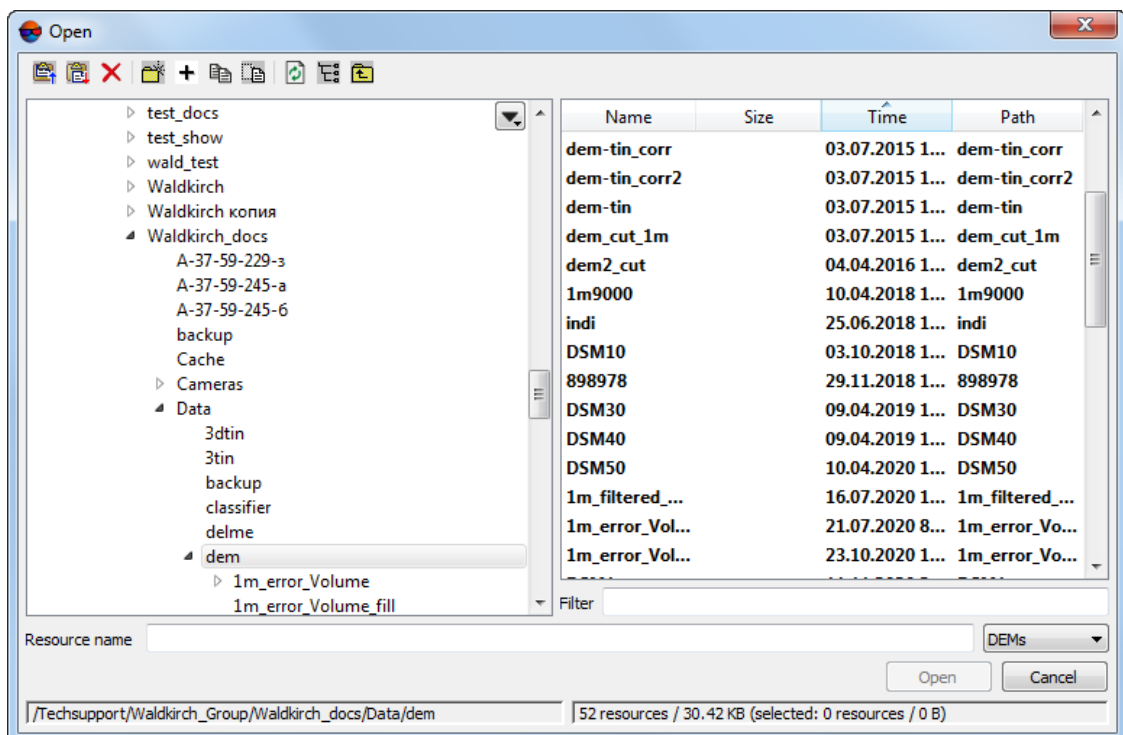


Fig. 222. DEM loading

2. Select one or multiple DEM files in active profile resources with *.x-dem extension.



When multiple files are selected, each DEM is loaded to separate layer.

3. Click **Open**. If one or multiple DEM containing layers are already loaded to active layer, then during loading of a new layer the window of loading type is opened.

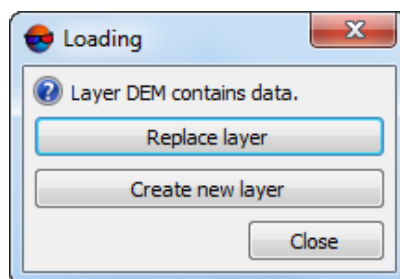


Fig. 223. DEM loading

The system provides the following possibilities of new layer loading to existing DEM layer:

- **Replace layer** – data of active layer is replaced by data from loading layer;
- **Create new layer** – data is loaded to a new layer.

Corresponding *DEM* layers are displayed in *Manager*.

The status line of 2D window displays cell height of active DEM as Z value.

To load DEM from external data without its conversion to internal format the **Terrain › DEM › Open from file...** menu item is used.



DEM loading and opening in format with extension different from *.x-dem may take a lot of time.

The system also allows to transfer lidar data to a DEM (see the '[LIDAR Data processing](#)' User Manual).

7.13. DEM export

7.13.1. DEM export

The system provides possibility of DEM export to the following formats.

- **ArcInfo ASCII grid** – exchange format with *.asc or *.grd extension, that is used in the program *ArcInfo*;

- **ASCII** – exchange text format with *.txt extension, that is supported by a large number of programs with different specializations (see description of the format in the '[Vectorization](#)') [User Manual](#);
- **SurferASCII grid** – exchange format with *.grd extension, that is used in the program *Surfer*;
- **USGS DEM** – exchange format with *.dem extension, developed in United States Geological Survey (USGS).

To export DEM, for example, to ArcInfo ASCII grid format choose **Terrain › DEM › Export › ArcInfo ASCII grid...**. The **Export to ArcInfo grid** window opens. Choose a folder to place a file in file system. Specify a file name in the **File name** input field. Click the **Save** button to complete the export.

7.13.2. Export to CSV

The system provides possibility of DEM export to CSV format. 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.

In order to export DEM to CSV format perform the following actions:

1. Select **Terrain › DEM › Export › CSV**. The **Export to CSV** window opens.
2. Choose a folder to place a file to file system.
3. Click the **Save** button. The **Export to CSV** window opens.

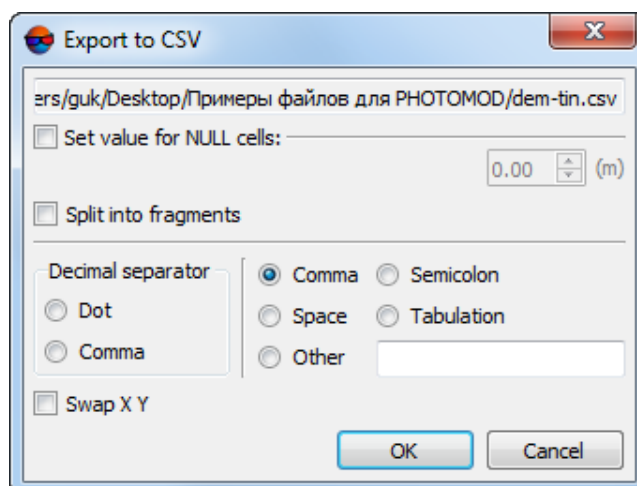


Fig. 224. Parameters of export to CSV format

4. [optional] To fill DEM null cells set the **Set value for null cells** checkbox on and input necessary value to the input field in meters.
5. [optional] To split CSV-file into separate parts, set the **Split into fragments** checkbox on.
6. In the **Decimal separator** section select point or comma to be used to separate coordinates.
7. In the **Field separator** section choose, what is used to separate fields: comma, space, tab, semicolon or other delimiters.



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 export operation.

8. [optional] In order to swap X,Y coordinates during DEM export, set the **Swap X Y** checkbox.
9. Click OK to complete the export operation.

7.13.3. Export to DTED

The system provides possibility of DEM export to Digital Terrain Elevation Data (DTED) format. This is exchange format with *.dt2 extension, that is used in the following programs: *ESRI ArcGIS Desktop* and *Pitney Bowes MapInfo*.

In order to export DEM to DTED format perform the following actions:

1. Select **Terrain > DEM > Export > DTED**. The **Export to DTED** window opens.
2. Choose a folder to place a file in file system.
3. Specify a file name in the **File name** input field.
4. Click the **Save** button. The **Options** window opens.

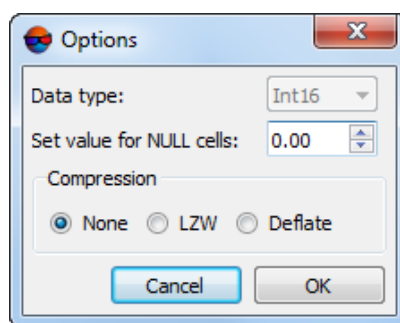


Fig. 225. Parameters of export to DTED format

To export the data type having an integer value in the **Data type** field the **Int16** parameters is set by default (integer data type is used to represent numbers in the range from -32 768 to 32 767).

5. [optional] To fill DEM null cells specify the **Set value for null cells** parameter in meters;
6. Set the **compression** parameters:
 - **None**;
 - The **LZW** compression – allows to use the Lempel-Ziv-Welch lossless data compression algorithm;
 - The **Deflate** compression – allows to use the Deflate lossless data compression algorithm.
7. Click OK to complete the export operation.

7.13.4. Export to ERDAS Imagine

The system provides possibility of DEM export to ERDAS Imagine format. This is exchange format with *.img extension, that is used in the following programs: *ERDAS Imagine* and *ESRI ArcGIS Desktop*.

In order to export DEM to ERDAS Imagine format perform the following actions:

1. Select **Terrain › DEM › Export › ERDAS Imagine**. The **Export to ERDAS Imagine** window opens.
2. Choose a folder to place a file in file system.
3. Specify a file name in the **File name** input field.
4. Click the **Save** button. The **Options** window opens.

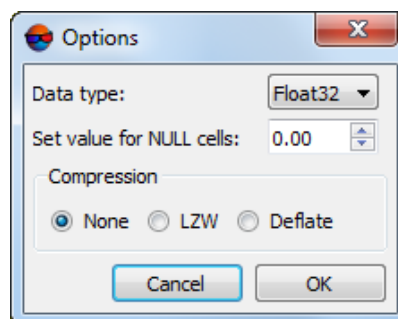


Fig. 226. Parameters of export to ERDAS Imagine format

5. Choose in the **Data type** list one of the following data types:
 - **UInt 8** – integer data type, that is used to represent numbers in the range from 0 to 255;
 - **Int 8** – integer data type, that is used to represent numbers in the range from -128 to 127;
 - **UInt 16** – integer data type, that is used to represent numbers in the range from 0 to 65535;
 - **Int 16** – integer data type, that is used to represent numbers in the range from -32,768 to 32,767;
 - **UInt 32** – integer data type, that is used to represent numbers in the range from 0 to 4,294,967,295;
 - **Int 32** – integer data type, that is used to represent numbers in the range from -2,147,483,648 to 2,147,483,647;
 - **Float 32** – data type, that is used to represent numbers with floating point in the range from 3.4e-038 to 3.4e+038;
 - **Float 64** – data type, that is used to represent numbers with floating point in the range from 1.7e-308 to 1.7e+308.
6. [optional] To fill DEM null cells specify the **Set value for null cells** parameter in meters.
7. Set the **compression** parameters:
 - **None**;
 - The **LZW** compression – allows to use the Lempel-Ziv-Welch lossless data compression algorithm;
 - The **Deflate** compression – allows to use the Deflate lossless data compression algorithm.
8. Click OK to complete the export operation.

7.13.5. Export to GeoTIFF

The system allows to export DEM to GeoTIFF format, that is exchange format with *.tif extension. GeoTIFF format – is common format for storing georeferenced images.

Information about used coordinates system for DEM are displayed as a code of European Petroleum Survey Group (EPSG) standard.

In order to export DEM to GeoTIFF format perform the following actions:

1. Select **Terrain › DEM › Export › GeoTIFF**. The **Export to GeoTIFF** window opens.
2. Choose a folder to place a file in file system.
3. Specify a file name in the **File name** input field.
4. Click the **Save** button. The **Export to GeoTIFF** window opens.

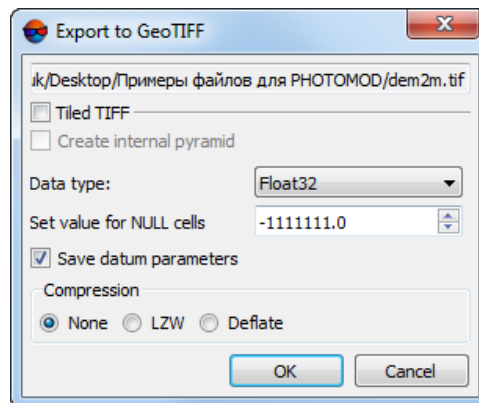


Fig. 227. Parameters of export to GeoTIFF format

5. Specify the following parameters:

- [optional] Set the **Tiled TIFF** checkbox to create tiles;



To set tiles size choose **Service › Settings**. The **Settings** window opens. In **System** tab set the **Tile size for writing MS-TIFF** (see the “System settings” chapter of the “[General system's parameters](#)” User Manual).



[optional] Set the **Create internal pyramid** checkbox to create pyramids for output image in files of internal format;

- Choose in the **Data type** list one of the following data types:
 - **UInt 8** – integer data type, that is used to represent numbers in the range from 0 to 255;
 - **Int 8** – integer data type, that is used to represent numbers in the range from -128 to 127;
 - **UInt 16** – integer data type, that is used to represent numbers in the range from 0 to 65535;

- **Int 16** – integer data type, that is used to represent numbers in the range from -32,768 to 32,767;
- **UInt 32** – integer data type, that is used to represent numbers in the range from 0 to 4,294,967,295;
- **Int 32** – integer data type, that is used to represent numbers in the range from -2,147,483,648 to 2,147,483,647;
- **Float 32** – data type, that is used to represent numbers with floating point in the range from 3.4e-038 to 3.4e+038;
- **Float 64** – data type, that is used to represent numbers with floating point in the range from 1.7e-308 to 1.7e+308.
- [optional] To fill DEM null cells specify the **Set value for null cells** parameter in meters.
- **Save datum parameters** – allows to save the appropriate additional parameters to metadata of TIFF-file;



To record this information, GeoTIFF tags with 34736 and 34737 codes are used (if the checkbox is cleared, this information is disabled and the default values are used – PHOTOMOD GeoReference).



In any case, the metadata include the coordinates of the DEM's upper left corner; the rotation matrix, including the cell size and coordinates of the upper left corner; the type of coordinate system; the units of the coordinate system; and the EPSG (GeoTIFF) code, if available.

- Set the **compression** parameters:
 - **None**;
 - The **LZW** compression – allows to use the Lempel-Ziv-Welch lossless data compression algorithm;
 - The **Deflate** compression – allows to use the Deflate lossless data compression algorithm.

6. Click OK to complete the export operation.

7.13.6. Export to PCIDSK

The system allows to export DEM to PCIDSK format, that is exchange format with *.pix extension and is used in the following program: *PCI Geomatics Geomatica*.

In order to export DEM to PCIDSK format perform the following actions:

1. Select **Terrain › DEM › Export › PCIDSK**. The **Export to PCIDSK** window opens.
2. Choose a folder to place a file.
3. Specify a file name in the **File name** input field.
4. Click the **Save** button. The **Options** window opens.

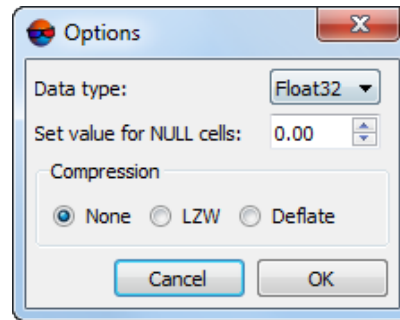


Fig. 228. Parameters of export to PCIDSK format

5. Choose in the **Data type** list one of the following data types:
 - **UInt 8** – integer data type, that is used to represent numbers in the range from 0 to 255;
 - **Int 8** – integer data type, that is used to represent numbers in the range from -128 to 127;
 - **UInt 16** – integer data type, that is used to represent numbers in the range from 0 to 65535;
 - **Int 16** – integer data type, that is used to represent numbers in the range from -32,768 to 32,767;
 - **UInt 32** – integer data type, that is used to represent numbers in the range from 0 to 4,294,967,295;
 - **Int 32** – integer data type, that is used to represent numbers in the range from -2,147,483,648 to 2,147,483,647;
 - **Float 32** – data type, that is used to represent numbers with floating point in the range from 3.4e-038 to 3.4e+038;
 - **Float 64** – data type, that is used to represent numbers with floating point in the range from 1.7e-308 to 1.7e+308.
6. [optional] To fill DEM null cells specify the **Set value for null cells** parameter in meters.

7. Set the **compression** parameters:

- **None**;
- The **LZW** compression – allows to use the Lempel-Ziv-Welch lossless data compression algorithm;
- The **Deflate** compression – allows to use the Deflate lossless data compression algorithm.

8. Click OK to complete the export operation.

7.13.7. Export to MTW

The system provides possibility of DEM export to the MTW formats. MTW is an exchange format with *.mtw extension that is used in the program *GIS Map 2000*.

In order to export DEM to MTW format perform the following actions:

1. Select **Terrain > DEM > Export > Panorama MTW...** The **Export to Panorama MTW** window opens.
2. Choose a folder to place a file in file system.
3. Specify a file name in the **File name** input field.
4. Click the **Save** button. The **Export to Panorama MTW** settings window opens.

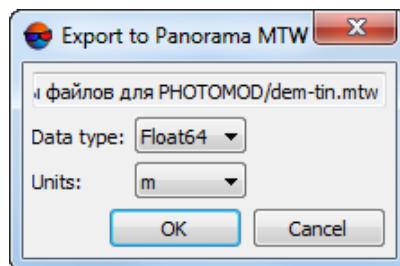


Fig. 229. Parameters of export to MTW format

5. Choose in the **Data type** list one of the following data types:

- **UInt 8** – integer data type, that is used to represent numbers in the range from 0 to 255;
- **Int 16** – integer data type, that is used to represent numbers in the range from -32,768 to 32,767;

- **Int 32** – integer data type, that is used to represent numbers in the range from -2,147,483,648 to 2,147,483,647;
- **Float 64** – data type, that is used to represent numbers with floating point in the range from 1.7e-308 to 1.7e+308.

6. Select DEM cell **Units** from the drop-down list.

7. Click OK to complete the export operation.

7.13.8. Batch DEM export

The system provides possibility to simultaneously export multiple DEM files of the same format.

To perform export of multiple DEM files do the following:

1. Select **Terrain > DEM > Export > Batch export**. The **DEM batch export** window opens.

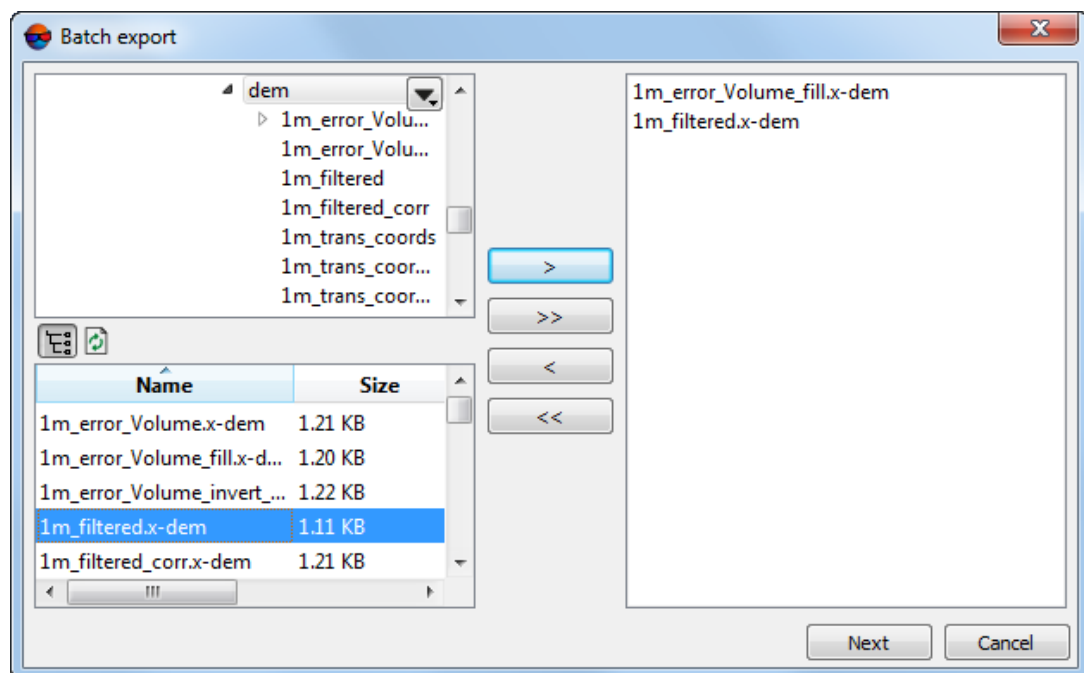


Fig. 230. Dem batch export

2. Choose a folder containing DEMs in resources tree.



 button allows to display all available resources in nested files.



 button allows to refresh part of the window with resources.



▼ button allows to show a list with 10 recently selected resources.

3. Select file with DEM in the list and click the ► button to add the file.



►► and ◀◀ allow to add to the list or remove from it all added files with vector objects, the ◀ button allows to remove selected file from the list.

4. Repeat the 2 and 3 actions to add next files with DEMs.
5. Click the **Next** button.

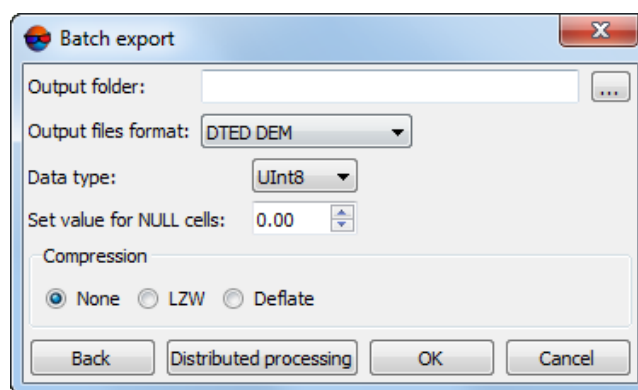


Fig. 231. Output folder and extension settings window

6. Click the ... button in the **Output folder** section and select a folder in file system.
7. Select format of exported DEM in the drop-down list.
8. Click OK to start operation of DEM export.

In order to perform batch DEM export using distributed processing, 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.

7.13.9. Convert DEM to point cloud (LAS)

The system provides for DEM conversion to LAS point cloud (see the “[LIDAR Data processing](#)” User Manual). For this, perform the following:

1. Choose **Terrain > DEM > Convert DEM to LAS....** The **Convert DEM to LAS** window opens:

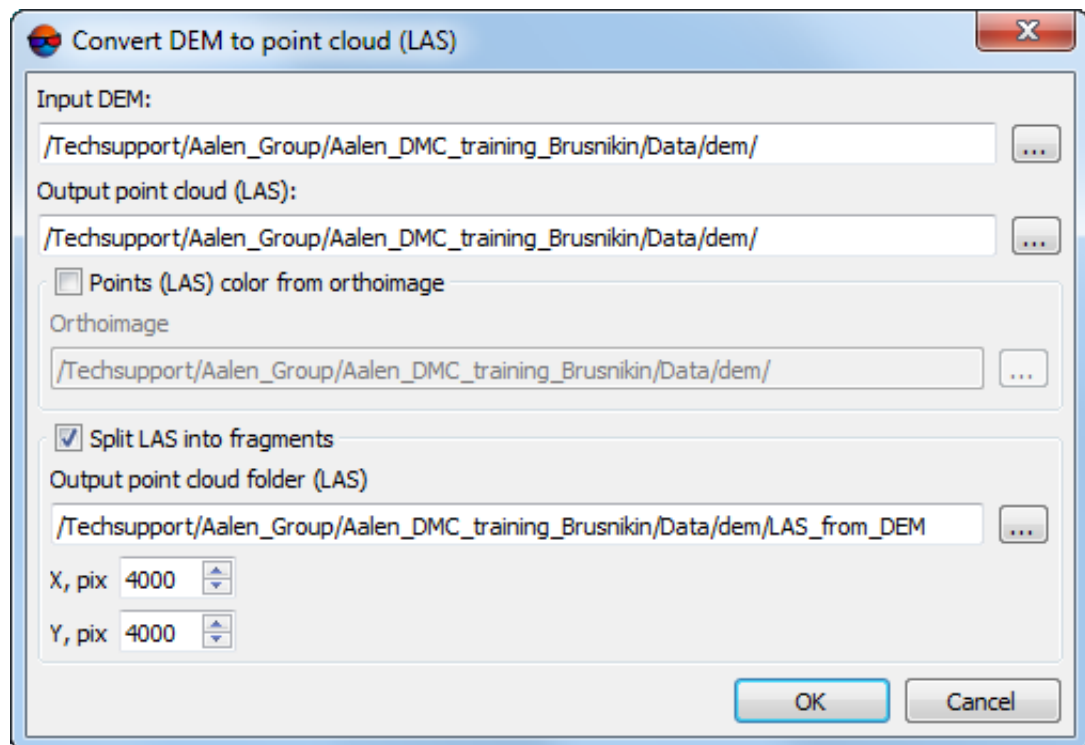





Fig. 232. The Export DEM to LAS window

2. Click the  button to select **Source DEM file path** in the active profile resources;
3. Click the  button to select **Output point cloud file path (LAS)** in the active profile resources;
4. To take **points (LAS) color from orthoimage**, set the appropriate checkbox. Click the  button to select the **orthoimage**;
5. In order to **Split points cloud on parts** set an appropriate checkbox, and specify the following parameters:
 - **Output point cloud folder (LAS)**;
 - **Part size by X, pix**;
 - **Part size by Y, pix**.
6. Click OK.

7.13.10. Convert DEM to point cloud (LAS) in batch mode

The system provides for batch DEMs conversion to LAS point clouds (see the “[LIDAR Data processing](#)” User Manual). For this, perform the following:

1. Select **Terrain > DEM > Batch conversion DEM to LAS...** The **Parameters – Step 1 of 2: Source data** window opens.

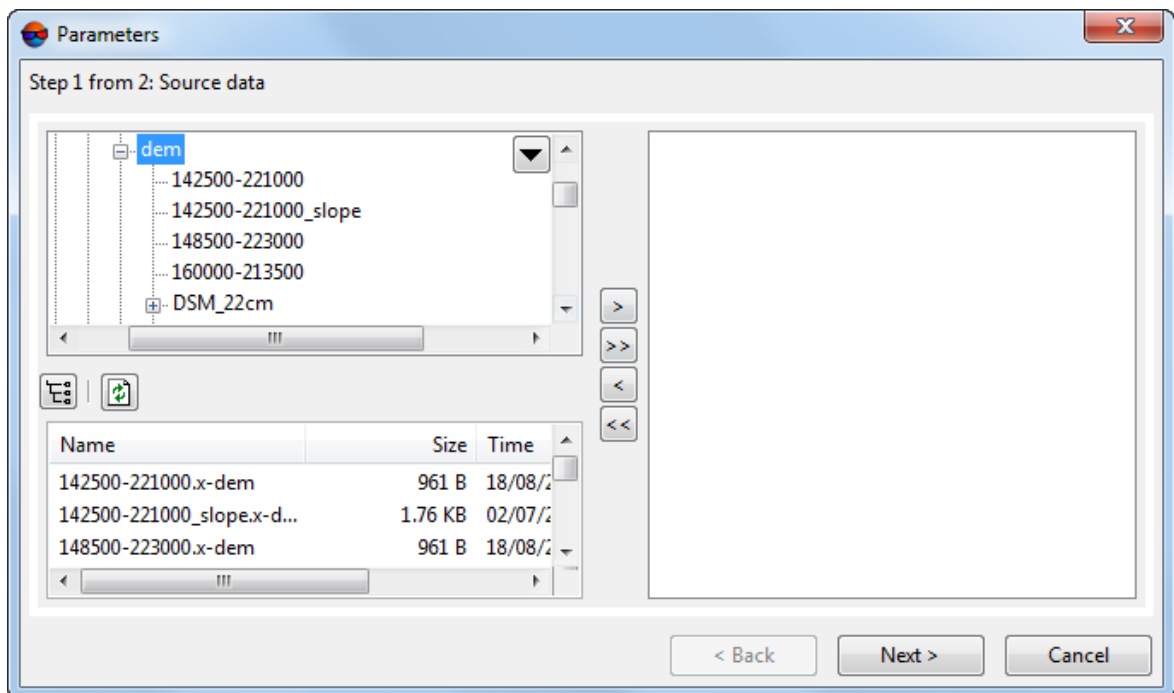








Fig. 233. Creating a list of the initial DEMs for batch conversion to LAS

2. Choose a folder containing DEMs in resources tree.

The  button allows to display all available resources in nested files. The  button allows to refresh part of the window with resources.

The  button allows to show a list with 10 recently selected resources.

3. Select DEM in the list and click the  button to add a layer.

The  and  buttons allow to add to the list or remove from it all added DEM files, the  button allows to remove selected file from the list

4. Repeat the 2 and 3 steps to add next files with DEMs.
5. Click the **Next** button. The **Parameters – Step 2 of 2: Parameters**.

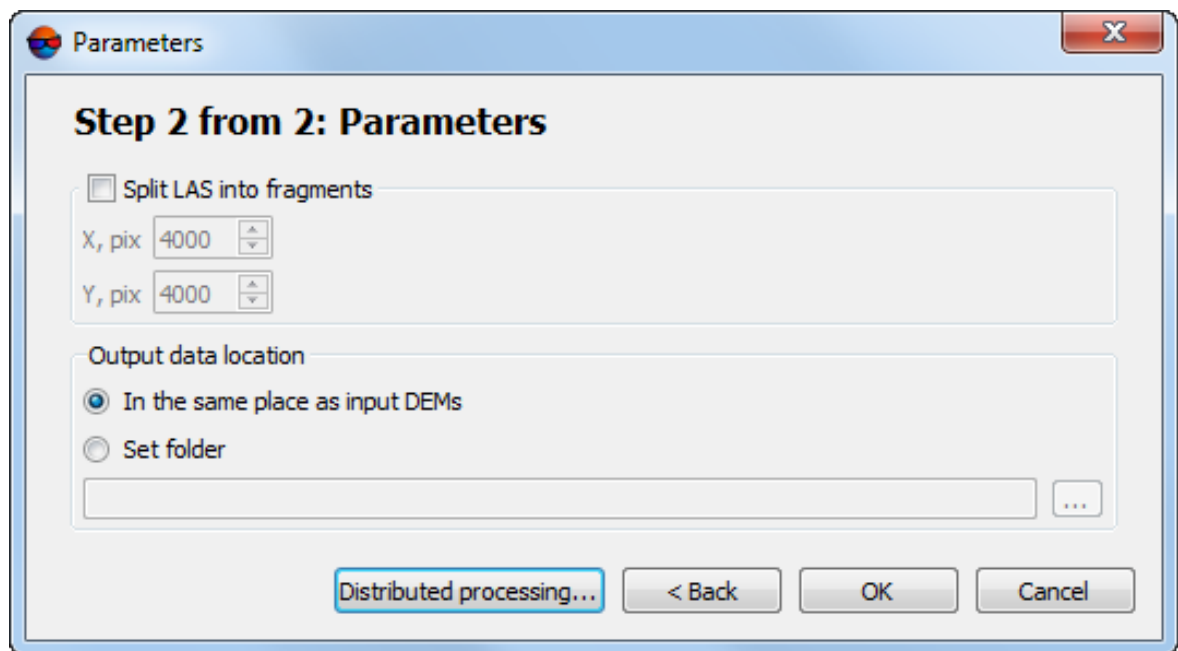



Fig. 234. Parameters of batch DTM conversion

6. Specify the following parameters:

1. Setup the **output LAS placement**:

- **In the same place as input DEM's;**
- Click the  button to **set the folder** for the output LAS files in the active profile resources.

2. [optional] In order to **Split LAS into fragments** set an appropriate checkbox, and specify the following parameters:

- Part size by **X, pix**;
- Part size by **Y, pix**.



If the **Split LAS into fragments** checkbox is cleared, all output LAS files will be saved in the indicated folder. Otherwise, fragments of point clouds will be distributed into subfolders with names that correspond to output DEM names.

7. Click **OK**.

To use the **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. Click the **Distributed processing** button.

7.14. DEM import

The system allows to import DEM from files with the following extensions: *.grd, *.asc, *.tif, *.dem, *.mtw, *.dt1, *.dt2, *.img, *.pix, *.hgt and *.xml.



For example – Japanese Fundamental Geospatial Data (FGD), which uses the Geography Markup Language Encoding Standard. The Geography Markup Language (GML) is an XML grammar for expressing geographical features.

Table 10. Brief description of the “Import” menu

Menu items	Function
Import...	contains items used to import DEM from files with *.grd, *.asc, *.tif, *.dem, *.mtw, *.dt1, *.dt2, *.img, *.pix, *.hgt extensions (see below)
Batch import	to simultaneous import of multiple DEM
Batch import from folder	to simultaneous import of multiple DEM from a specified folder

In order to perform import of DEM from a file of supported format, do the following:

1. Select **Terrain > DEM > Import > Import**. The **DEM import** window opens.
2. Select a DEM file.
3. Click the **Open** button. The **DEM import** window opens.

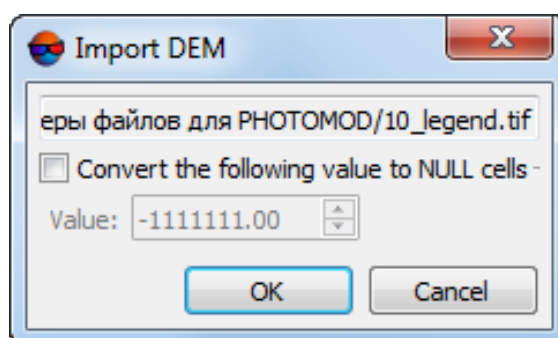


Fig. 235. Parameters of DEM import

4. [optional] To convert DEM cells with specified values to null cells, set the **Convert the following value to null cells** checkbox and define necessary value.

- Click OK. The **Destination file** window opens.

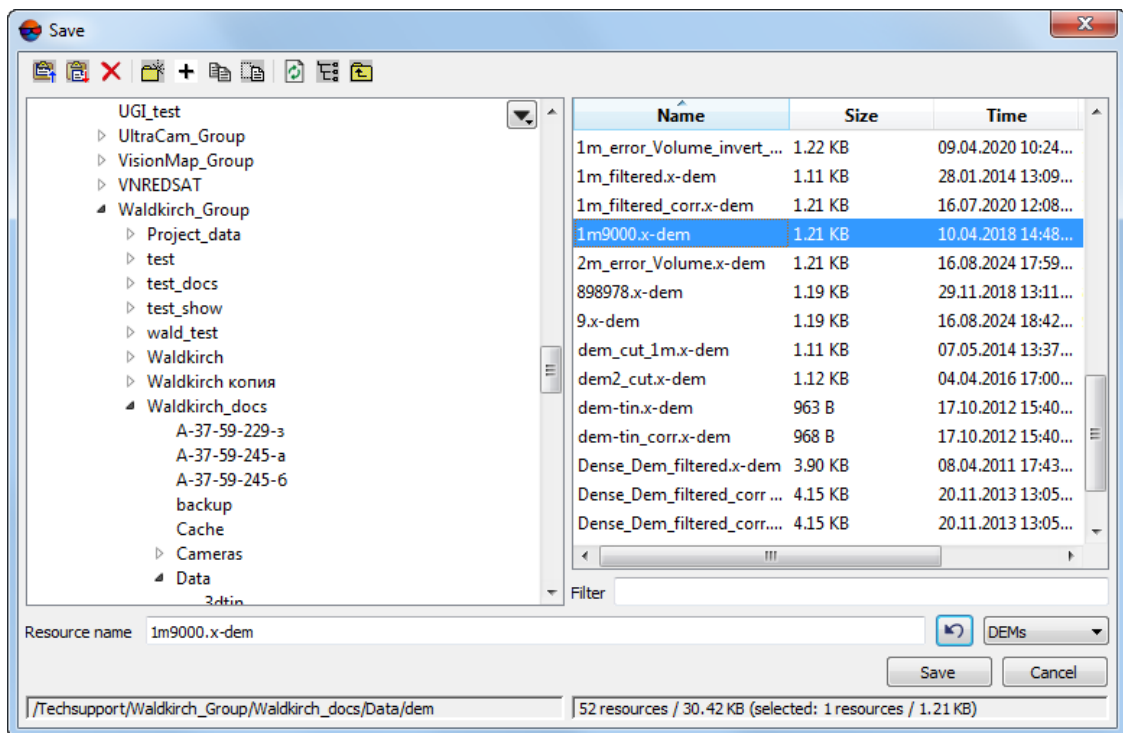


Fig. 236. Imported DEM saving

- Select target file placement in active profile resources and click **Save**.

7.14.1. Batch DEM import

The system allows to import DEM in batch mode from files with the following extensions: *.grd, *.asc, *.tif, *.dem, *.mtw, *.dt1, *.dt2, *.img, *.pix, *.hgt. During batch import the system loads multiple files from specified folder at once.

In order to import necessary number of DEMs, from specified folder, perform the following actions:

- Select **Terrain > DEM > Import > Batch import**. The **DEM import** window opens.
- Select multiple files with DEM.
- Click the **Open** button. The **DEM import** window opens.

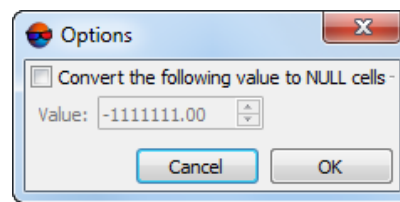


Fig. 237. Parameters of DEM import

4. [optional] To convert DEM cells with specified values to null cells, set the **Convert the following value to null cells** checkbox and define necessary value.
5. Click OK. The **Select folder** window opens.

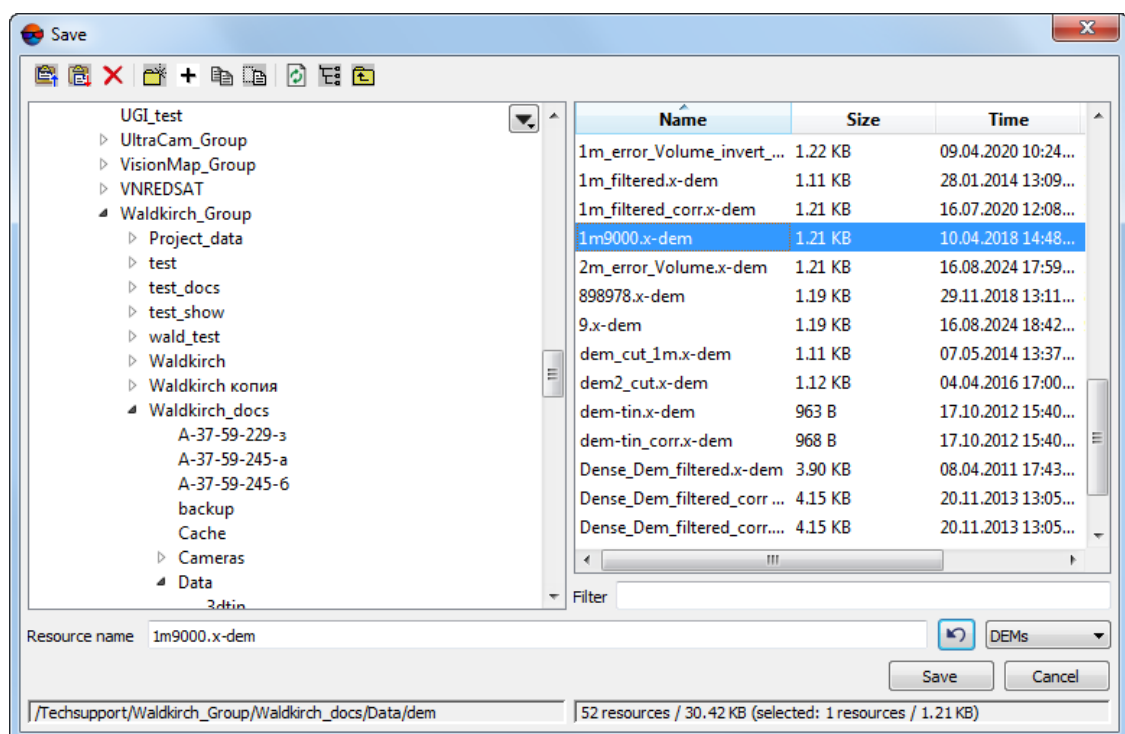


Fig. 238. Imported DEM saving

6. Select a folder or create a new one using the **Create...** button.
7. Click OK to complete the import operation.

7.14.2. Batch import from folder

The system allows to import DEMs from the folder specified in a batch mode. While using a batch import from the folder all the DEM files with *.grd, *.asc, *.tif, *.dem, *.mtw, *.dt1, *.dt2, *.img, *.pix, *.hgt format extensions are loaded.

To import all the DEMs placed in a specified folder, perform the following actions:

1. Choose **Terrain > DEM > Import > Batch import from folder....** The **Files searching by filter** window opens;

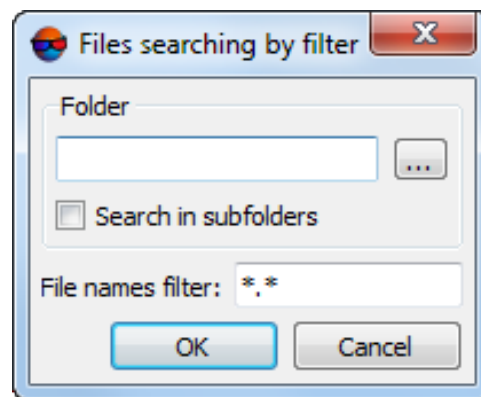



Fig. 239. Files searching by filter window

2. Click  button to setup target folder for import in file system;
3. [optional] turn on **Search in subfolders** checkbox to import DEMs from the sub-folders;
4. [optional] if it's needed to import just some of the DEMs placed in the folder specified, in **File names filter** setup parameters of sorting DEMs by names and/or by extensions;



In default, all the DEMs placed in the folder specified will be imported.

5. Click OK. The **Parameters** window opens;

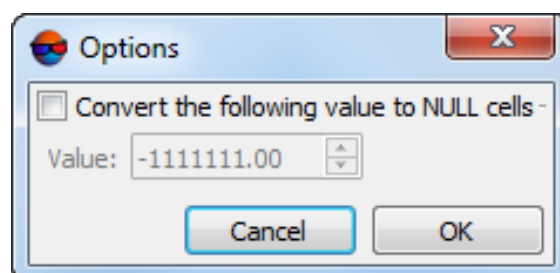


Fig. 240. Parameters of importing DEMs

6. [optional] To convert DEM cells with specified value to NULL cells, turn on **Convert the following value to NULL cells** checkbox and setup a value.

7. Click OK. The **Select folder** window opens.

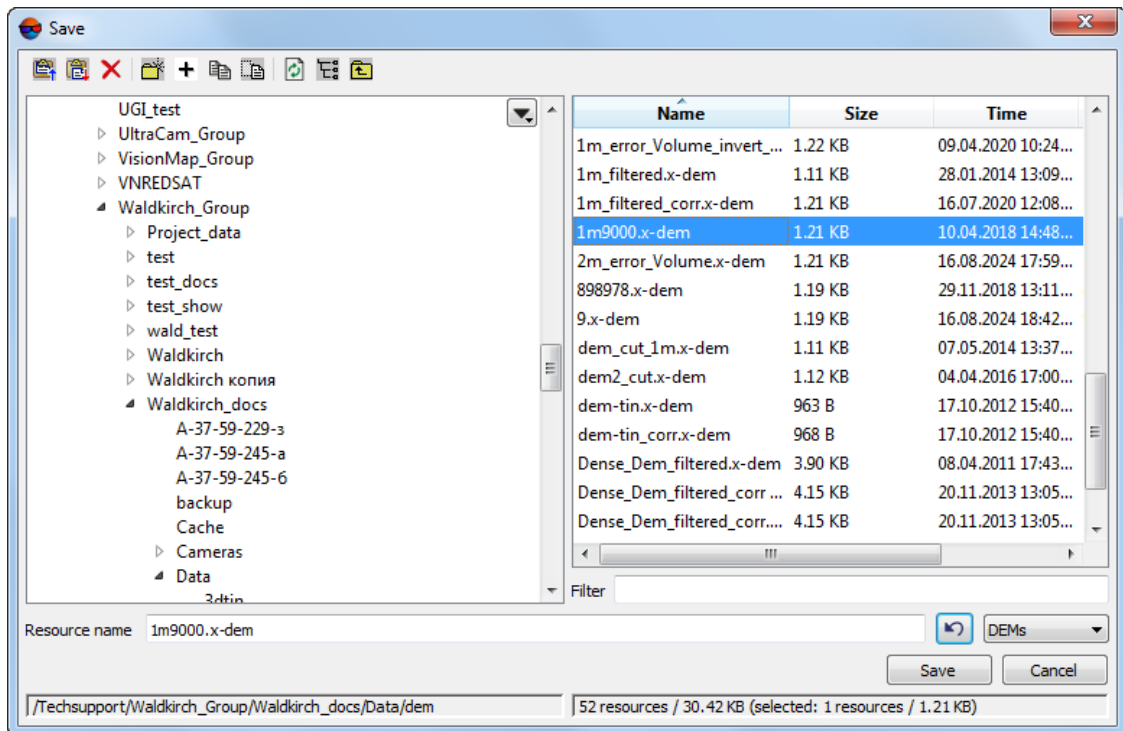


Fig. 241. Imported DEM saving

8. Select the folder or create a new one with **New...** button.

9. Click OK to finish import.

8. Volumes Calculation

The system provides opportunity to calculate a volume (i.e. *embankment* or *excavation*) located between DEM (or TIN) surface and *arbitrary Z-plane* or in a more complex case, the volume which is the *overlap* between the two (roughly, “top” and “bottom”) surfaces, which can intersect or not.



When calculating two volumes constrained by the same surface represented either by TIN or DEM (e.g. build using this TIN) the system achieves the accuracy wherein the difference between the calculated volumes (by DEM and by TIN) is no worse than 1%.

The principle of calculation method comes to calculation of DEM cells volumes (cells should be with known cell corners coordinates, otherwise, the cell is missed and its volume is not considered). And the surface corresponding to the “ceiling” of the cell is built by linear interpolation from the corner points.



The system also provides volume calculation if DEMs have different ground sample distance.

To calculate a volume perform the following actions:

1. Choose **Terrain > Compute volumes**. The **Volumes calculation** window opens;

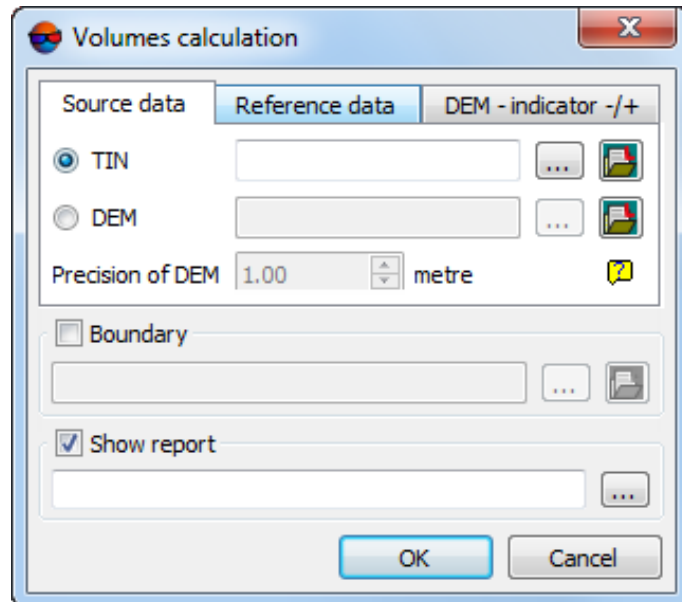



Fig. 242. The Volumes calculation window, Source data tab

2. In the **Source data** tab click an appropriate  button to select the **TIN** or **DEM** surface, bounding volume to be calculated;



Click the  button to open the **Layer selection** window and select the source DEM (or TIN) from the list of the appropriate layers uploaded to the project.

3. [optional] In the **Source data** tab specify the **Precision of DEM** in meters – a priori error of elevations.



For *DEM-indicator* cells with a lower precision a volume of changes equals null.

4. In the **Reference data** tab specify constant height of plane, bounding calculated volume “from below” or select another TIN or DEM replacing the first one (in case of calculating a volume obtained by surfaces overlap);

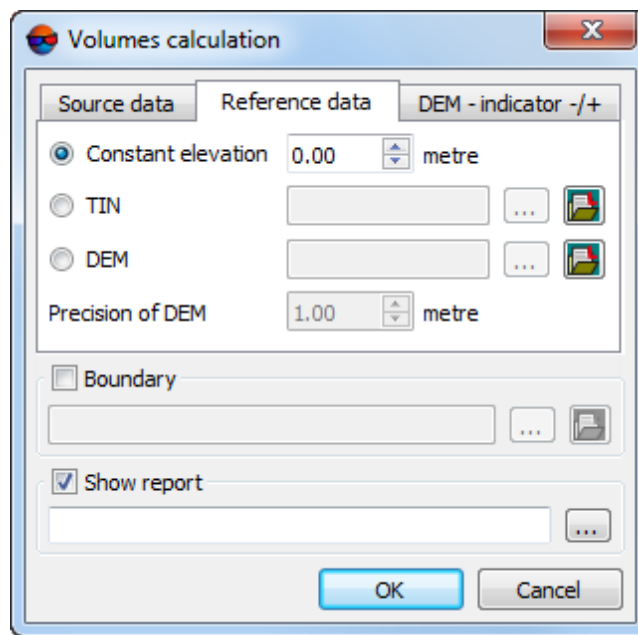


Fig. 243. The Volumes calculation window, Reference data tab

5. [optional] Specify the **Precision of DEM** in meters in the **Reference data** tab;
6. [optional] To create referential DEM, indicating embankments and excavations, set the **DEM – indicator +/-** checkbox in the **DEM – indicator +/-** tab and specify path to save a file with output referential DEM.

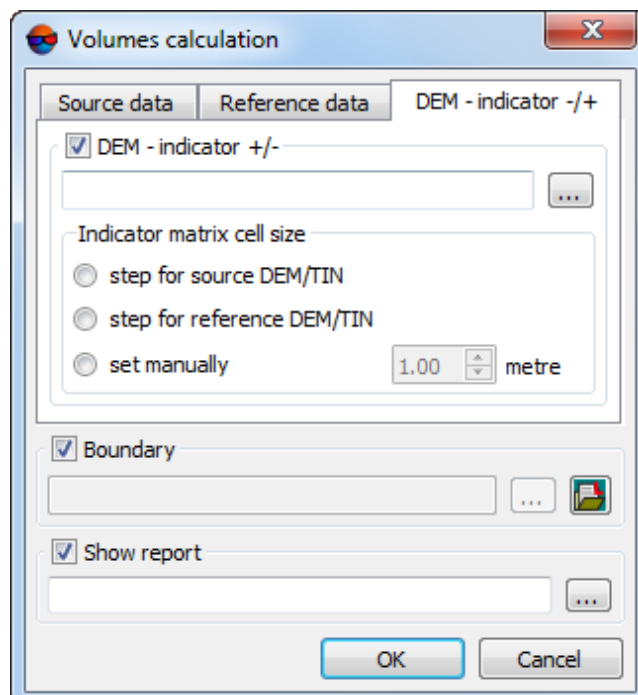


Fig. 244. The Volumes calculation window, DEM – indicator tab

Specify the **Cell size for indicator DEM**:


- **Step from source DEM/TIN;**
- **Step from reference DEM/TIN;**
- specify the cell size manually.



“Step” from source/reference DEM – DEM cell size.




“Step” from source/reference TIN – average distance between vertices of triangulation irregular network.


7. [optional] Set the **Boundary** checkbox and click the  button to specify a boundary, additionally bounding calculated volume by XY.



Boundary of calculated volume may be additionally bounded *only* by vector layer (polygon) in *PHOTOMOD* format. Otherwise, it will be necessary to convert used data to *PHOTOMOD* format.



Click the  button to open the **Layer selection** window and select the vector layer from the list of the appropriate layers uploaded to the project.

8. [optional] Set the **Show report** checkbox and click the  button to specify path to save a report file with volume calculation data.
9. Click OK.

The **Report** window opens when the calculation is complete.

Volume computing results

PHOTOMOD version 8.0.4639 x64

Project properties and initial data

Project	
Report time	28 августа 2024 г., 14:54:58
Coordinate system	Cartesian Right
Height range,	0.000 - 0.000
Number of images	0
Number of strips	0

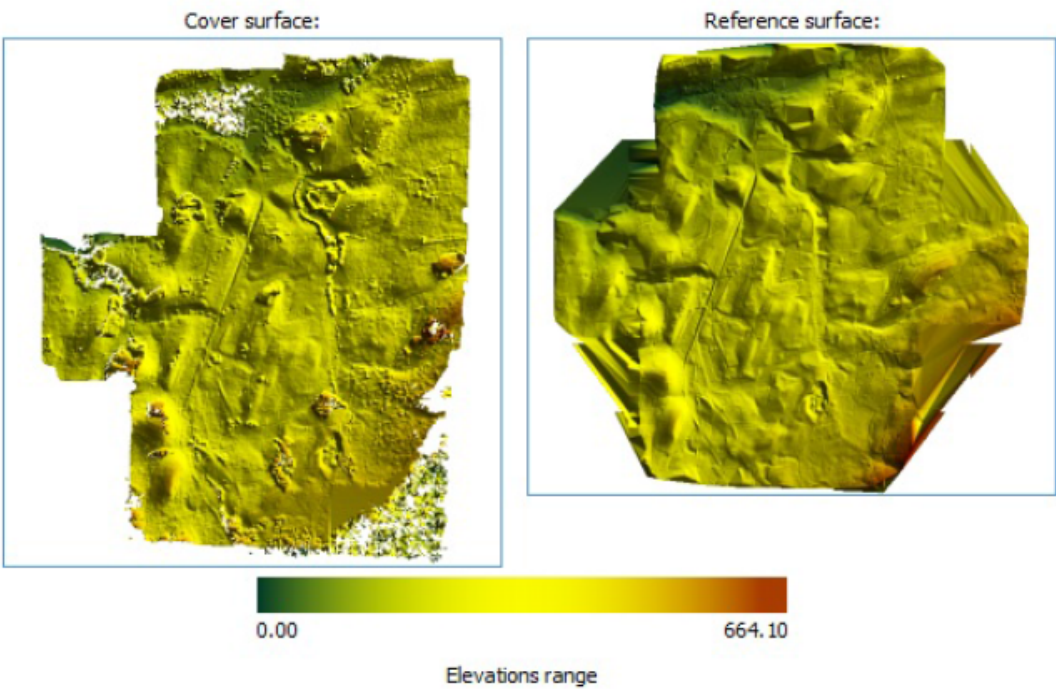


Fig. 245. Report with volume calculation results (reference data – Z-plane with constant height)






Volume computing results	
Property	Value
Embankment volume	0.000 m ³
Excavation volume	0.000 m ³
Relative volume of changes	0.000 m ³
Absolute volume of changes	0.000 m ³
Target area	0.000 m ²
Prior error estimate	±0.000 m ³
Embankment volume error	±0.000 m ³
Excavation volume error	±0.000 m ³
Failed cells	0
Skipped cells	0
Precision for Z	±1.000 m


Cover surface:	
File	\\Techsupport\\Waldkirch_Group\\Waldkirch_docs\\Data\\dem\\final_copy.x-dem
Width	3 352 pixels
Height	3 557 pixels
Step	1.000 m × 1.000 m
Precision of Z axis	± 1.000 m

Reference surface:	
File	\\Techsupport\\Waldkirch_Group\\Waldkirch_docs\\Data\\dem\\Dense_Dem_final.x-dem
Width	3 984 pixels
Height	3 451 pixels
Step	1.000 m × 1.000 m
Precision of Z axis	± 1.000 m

Fig. 246. Report with volume calculation results (reference data – DEM)

Report contains the following main elements and sections:

- the toolbar with buttons used to perform the following operations:
 -  – allows to return to previous view of the report;
 -  – allows to refresh results in the report;
 -  – allows to perform a search for information in the report;
 -  – allows to save report data to files with *.htm, *.html extensions out of active profile resources;
 -  – allows to save report data to files with *.htm, *.html extensions in active profile resources;

-  – allows to print a report.
- the section of a report containing brief project properties:
 - a project name;
 - report generating date;
 - project coordinate system.
- the section of a report containing calculation results:
 - **Embankment volume**;
 - **Excavation volume**;
 - **Relative volume of changes** – math difference between embankment volume and excavation volume;
 - **Absolute volume of changes** – the sum of embankment and excavation volumes;
 - The area where calculations were made;
 - Information about the surface, bounding volume to be calculated;
 - Information about the plane with constant height (or DEM), bounding calculated volume “from below”;
 - Error a priori estimate.