USER MANUAL

TABLE OF CONTENT

1 INTRODUCTION

Dharohar is an application for non-commercial use in damage detection and cultural heritage developed at the Indian Institute of Remote Sensing (IIRS), Indian Space Research Organization (ISRO), Dehradun India.

The purpose of this project is to handle the problem related to damage detection. In this Project we are developing a software i.e. Dharohar which is a customised version of various software performing different functionalities of image processing. We provide point cloud(3-D image) as input then we perform various operations simultaneously like translate, rotate, unroll, rasterize, co-registration, edge detection, change, detection, rasterization. After visualisation feature extraction is performed which includes energy, entropy, homogeneity, correlation, contrast etc. This process is done by using GLCM (grey level co-occurrences matrix) texture analysis method. Apart from visualisation and texture analysis this project also include database part which is used for storage and querying.

2 GENERAL CONCEPTS

2.1 EXECUTION OF SOFTWARE:

- Execute the give .exe file 'Dharohar.exe' to run the application.
- User can select the preferable language from the window that appears as follows:

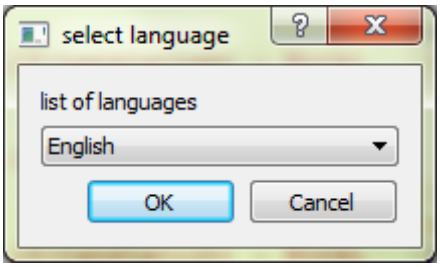


Fig 1: Language selection window

Selection of the language will change the entire GUI to the corresponding language.

- After language setting the window appears as follows:

Fig 2: Main Window

2.2 ADMINISTRATOR LOGIN:

- All the necessary details in the Main Window have to be filled for authentication. After authentication a new window appears as follows:

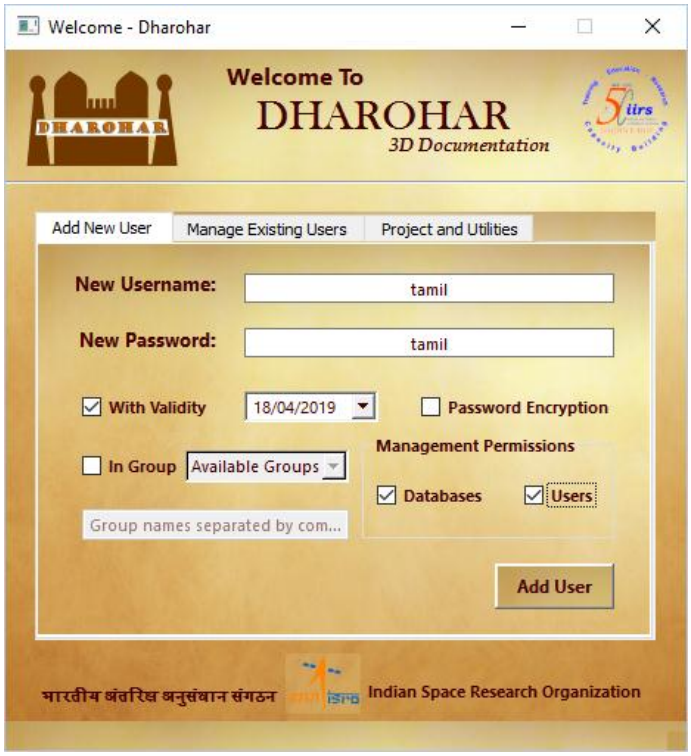


Fig 3: Add new user

- Administrator can add new user and create password for the user set validity and can enable password encryption.
- Administrator can also add the user to Group if there is any available group.
- Administrator can provide management permission to the user.
- If the Administrator does not provide permission for user. Then the user can not be able to work on the project.
- Administrator can also manage the existing user and can change password for the user.
- Can also update the management permission ,validity for the user
- And can also delete the user.

The name “Indian Space Research Organisation” at the bottom of the window has a link directed to the website in English and Hindi name directs to the website in Hindi

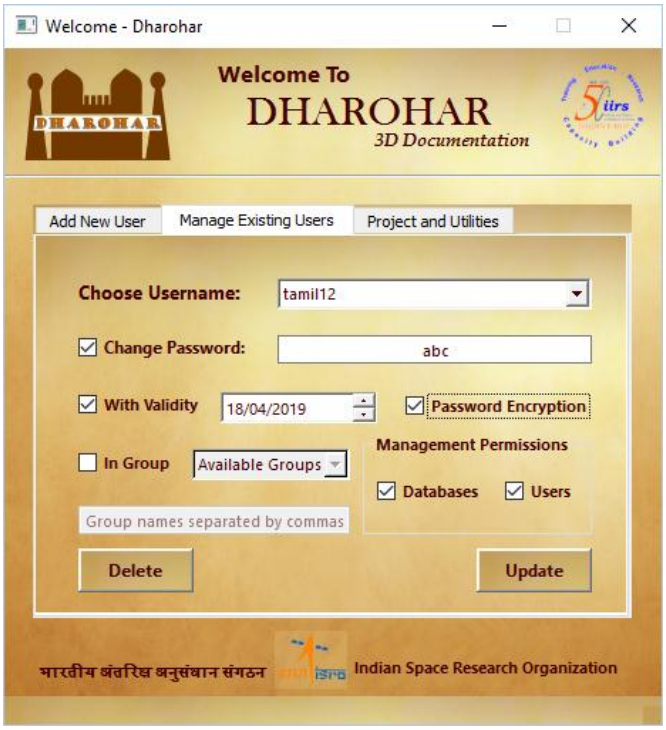


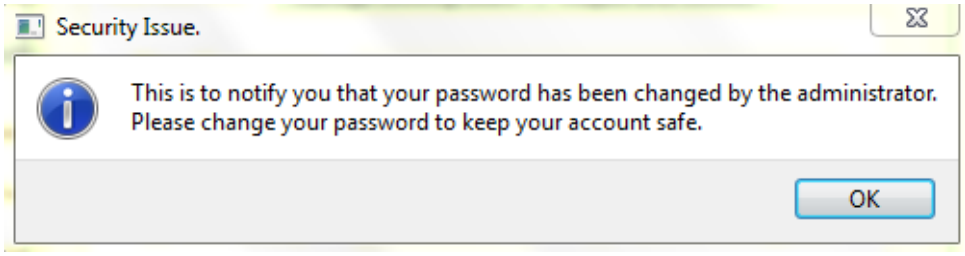
Fig 4: Manage Existing user

- The administrator can also manage the project for the user.
- But the choosing the user and selecting either insert or select or both the administrator can provide privileges for the user.
- Only the administrator can delete the projects created.

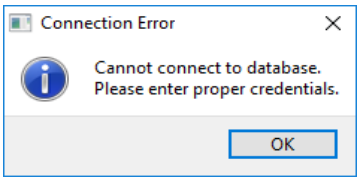
Fig 5: Manage project

2.3 USER LOGIN

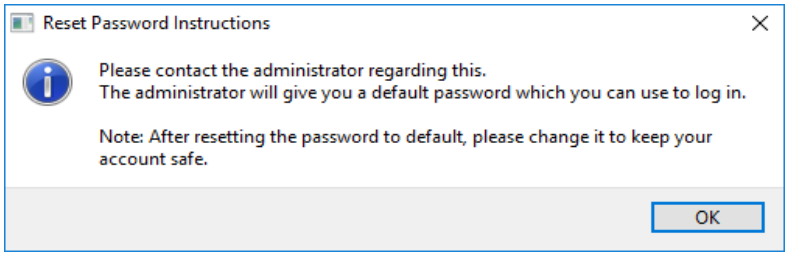
- In case the new user authenticate himself and login with the password created by the administrator for the first time a message box appears insisting the user to change the password.



- In case of unauthorized user a window with error message appears.



- If the user forgets his/her password then they can click on forgot password button and a message box appears as follows:



- The user is allowed to proceed to the next page where he/she can change his/her own password by clicking on the *Change Password* button.

Fig 6: Project selection

- The default created by the administrator has to be provided as old password.

Fig 7: Change password

- The *Add New User* and *Manage Existing User* tabs get enabled only if the user is an administrator.

Credits:

Contact us:

Help:

Help menu opens a pdf file of the complete user manual of "DHAROHAR".

Logout:

Logout button logs out the user and comes back to the authentication page for login in again.

2.4 PROJECT SELECTION:

- The user can choose any existing project from the drop-down list from *choose project* or else the
- User can also create New Project by giving the Project Name.
- After selecting the project the user has to click on proceed button which takes the user a new page based on his/her permission given by the administrator.

Fig 8: Processing window

- The project name and the user name who has been logged in will be displayed at the top left corner of the window.

3. LIST OF PROCESS:

I. POINT CLOUD PROCESSING
II. LOAD IMAGE LIST FROM DATABASE.
III. IMAGE ANALYSIS
IV. CHANGE DETECTION
V. TEXTURE ANALYSIS
VI. POINT ATTRIBUTES.

3.1 POINT CLOUD PROCESSING :

- Click on *point cloud processing* button to access this tool.

3.1.1 Graphical User Interface:

Here is a quick overview of the main user interface. The main user interface contains the Project Name that the user is currently working on, The User Name, Password, Location, Database Name and the Port Number.

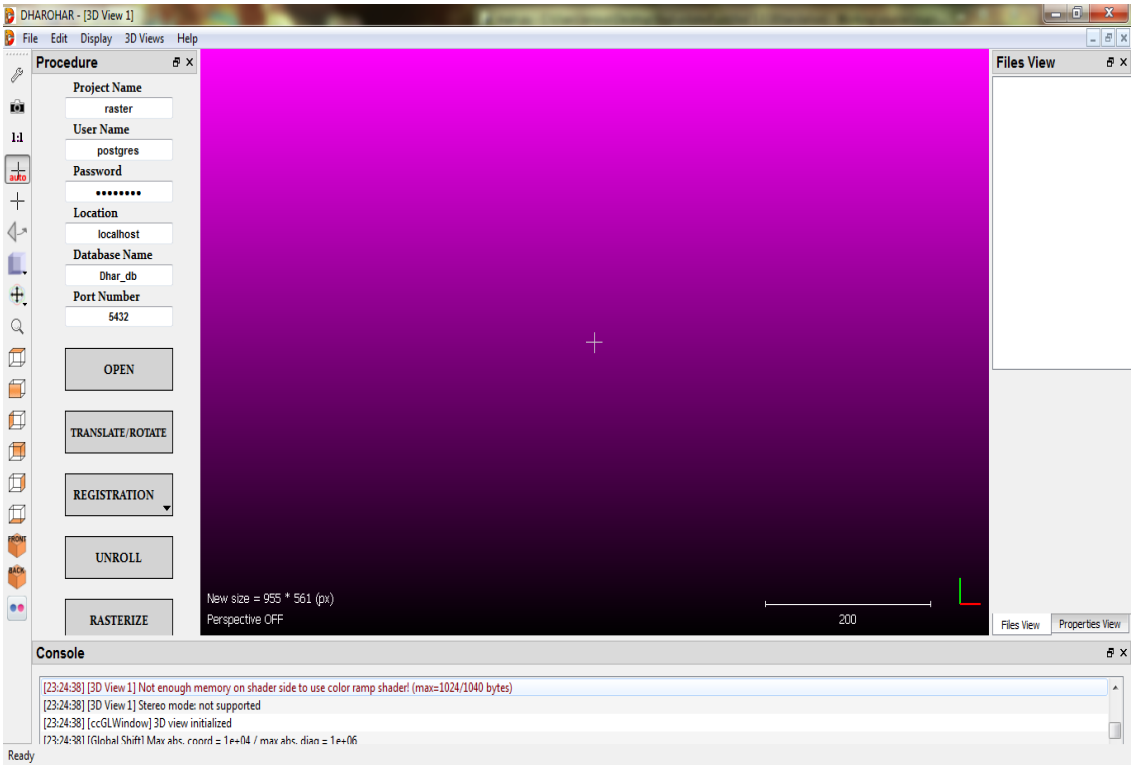


Fig 9: Point cloud processing

I .Procedure:

- Open (open the point cloud)
- Translate\Rotate
- Registration
  - Reference Image
  - Align point pair Registration
  - Fine Registration
- Unroll
- Rasterize

II. Menu:

- File (open, open recent, save, close all, quit…)
- Display (full screen, refresh, and reset all GUI elements…)
- 3D views (new, close, zoom in, zoom out...)
- Help (help, about, Enable qt warning in console…)

II. View toolbar (quick access to display-related tools)

IV. Property view (information on selected entity)

V. File view (list of selected entity)

VI. Default 3D view

VII. Another 3D view (created with 3D Views > New)

VIII. Console.

3.1.2 Entities:

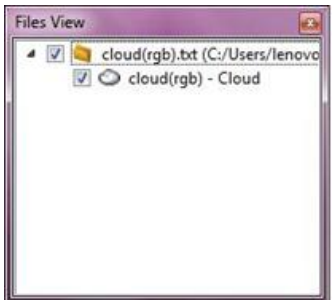


Fig 10: File view with entities

I .POINT CLOUD:

A point cloud is a set of unorganized 3D points (X, Y, and Z).

It can be associated to:

- A unique colour for the whole entity (RGB)
- Per-point colours (RGB)
- Per-point normal vectors (Nx, Ny, Nz)
- Per-point scalar values (a scalar field) – multiple scalar fields can be associated to the same cloud

II. PREMITIVES:

- Primitives are a special kind of meshes. They can be created with the 'Primitive Factory', or with the 'Tools > Fit' methods (or also imported from CAD formats – e.g. PDMS macros).
- Primitives are described by simple parameters (radius, height, etc.). However they are associated to a tessellated representation (i.e. a proper triangular mesh). This way they can be used as standard meshes (for distance calculation, etc.).

Note: for some primitives (spheres, cylinders, etc.) the user can change the 'drawing precision' (i.e. the amount of tessellated triangles).

III. DB TREE:

Loaded entities are all stored in the 'DB tree' (on the left part by default). Some entities can depend on other ones (such as a mesh and its vertices) or can also be regrouped (in Group entities). This is why the database is displayed as a hierarchical tree.

Selection:

Entities can be selected either directly in a 3D view (by left clicking on it) or by clicking on their corresponding entry in the DB tree (which is generally faster and unambiguous).

Multiple entities can be selected at once by maintaining the CTRL or SHIFT keys pressed and selecting them in the DB tree.

Equivalently the user can select multiple entities by holding the CTRL key and clicking on entities in the 3D views. Another option is to hold the ALT key and drawing a rectangle in a 3D view. This way, all the entities falling at least partly inside the rectangle will be selected:

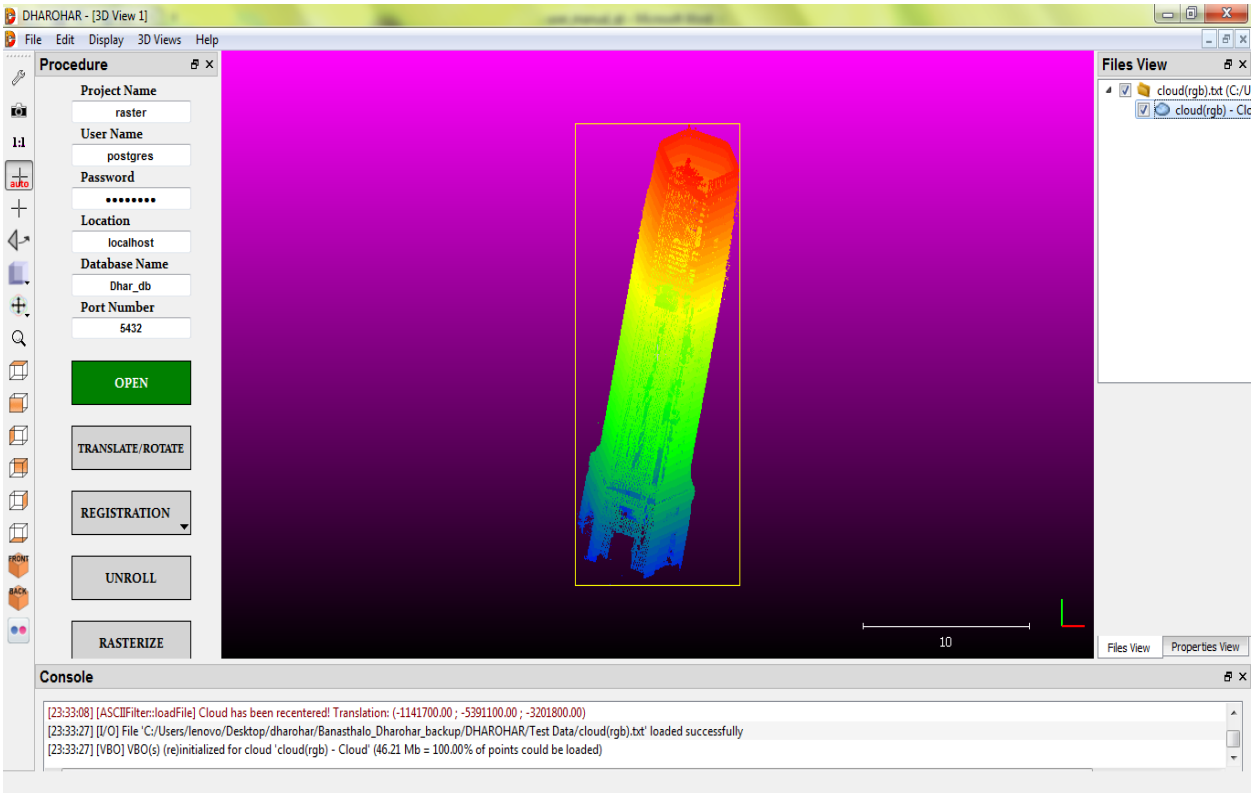


Fig 11: Selected point cloud

3.1.3 Procedure for processing

(A) OPENING OF POINT CLOUD:

- Click on Open in main menu to open the point cloud.
- A window with the list of files from your system drive will appear (as in Fig 2).
- Navigate to the folder in which the point cloud data is present and select the point cloud you want to open and click open.

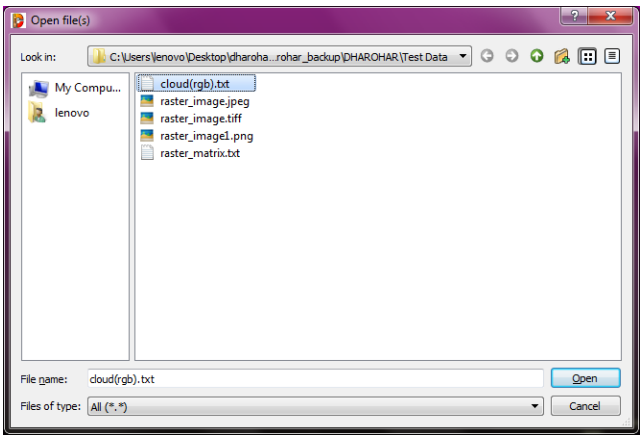


Fig-12: Opening Point Cloud

- A window with ASCII files will appear (as in Fig 3).
- Click on apply button to open the point cloud.

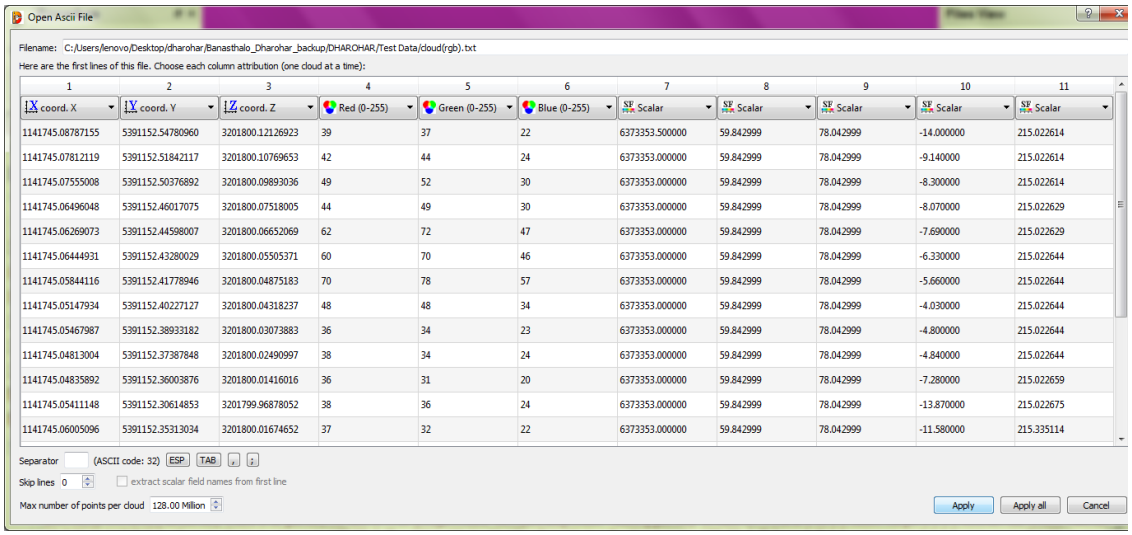


Fig 13: Applying ASCII File

- Click “Yes” to affine the global shift of the coordinates.

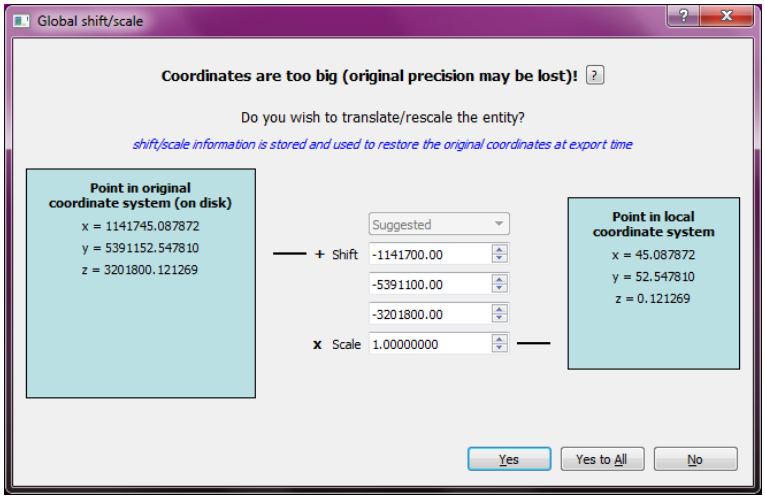


Fig 14: Co. Ordinates Affirmation

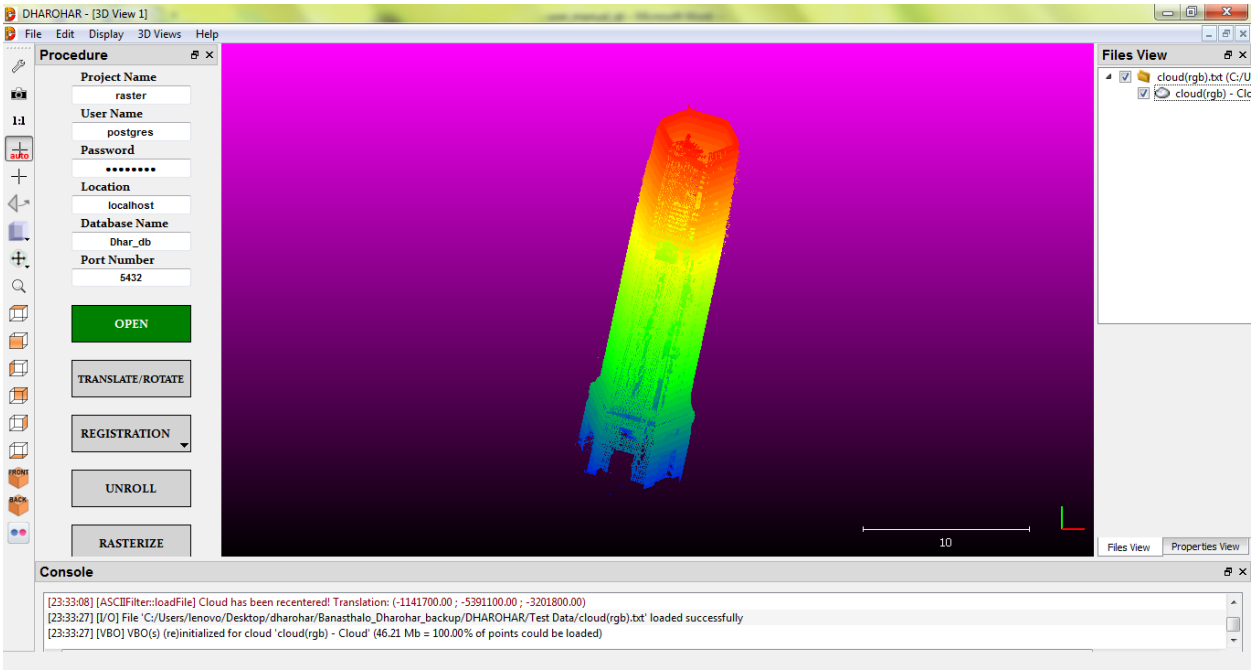


Fig 15: Opened Point Cloud

- The Loaded entities are all stored in the DB Tree and are displayed in the File view. Some entity can depend on the other entity and can also be regrouped so they are displayed as hierarchical tree.
- At the bottom right corner of the main window the coordinates of the opened point cloud is shown. Green line indicates y-axis, Red line indicates z-axis and Blue line indicates x-axis.

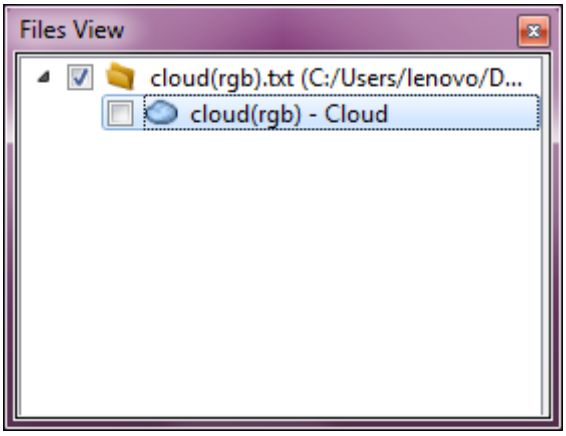


Fig 16: File view with entities

(B) TRANSLATION/ROTATION:

Description

This tool allows the user to interactively move the selected entities relatively to the other ones (or equivalently to the default coordinates system).

Start

The user must select one or several entities before launching this tool. The entities can be any 3D geometry entity (clouds, meshes, polylines, primitives, etc.).

- Notes: locked entities (sub-meshes, etc.) can't be moved this way
- only the entities displayed in the active 3D view will be considered

Procedure

The standard mouse interactions with the 3D view are used to modify the selected entities position (instead of the current camera):

- left click: rotate
- right click: translate

Pause

At any time, the user can 'pause' the transformation mode (click on the 'pause' button or hit the space bar) in order to modify the camera position/orientation, and then restart the transformation process by 'un-pausing' it (new click on the 'pause' button or new hit on the space bar).

Constraints

Optionally, constraints can be added to the applied transformation:

- Rotation can be constrained to a single axis (X, Y or Z). Use the drop-down menu to select the current dimension.
- Translation ca be constrained to zero, one or two dimensions only (among X, Y and Z). Just uncheck the dimensions that should be ignored.

When done, use the validation icon to apply the transformation, or the cancel button to revert it.

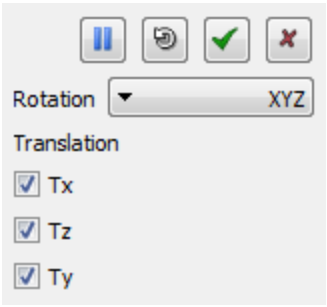


Fig 17: Translation / Rotation

(C) REGISTRATION/GEO-REFERENCING:

- This tool lets the user align two entities by picking at least three equivalent point pairs in both entities.
- This method is very useful to align clouds quite precisely. It's even sometimes the only way to get a fine result (typically if the two clouds have great differences on large extents, in which case the ICP registration won't work properly).

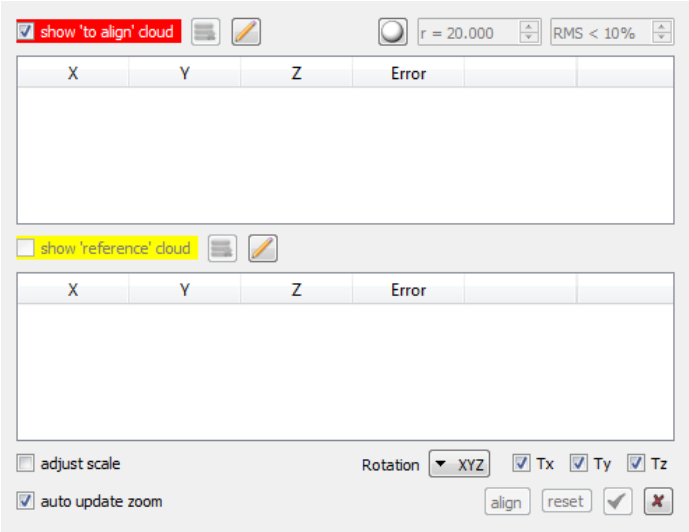


Fig 18: Registration Window

Reference Image:

Open reference image for aligning the point cloud.

Align point pair registration:

- Select both the reference and the align cloud from the db tree
- Click align point pair registration button from the main menu.

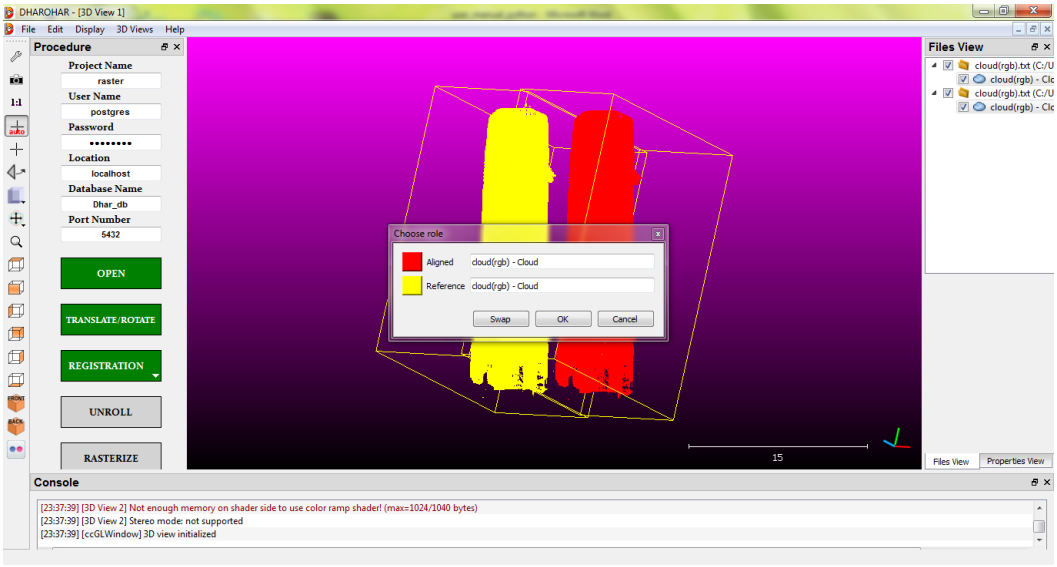


Fig 19: align point pair

- Pick the points from the align cloud and its corresponding reference point from the reference image (minimum of 3 points are to be picked).

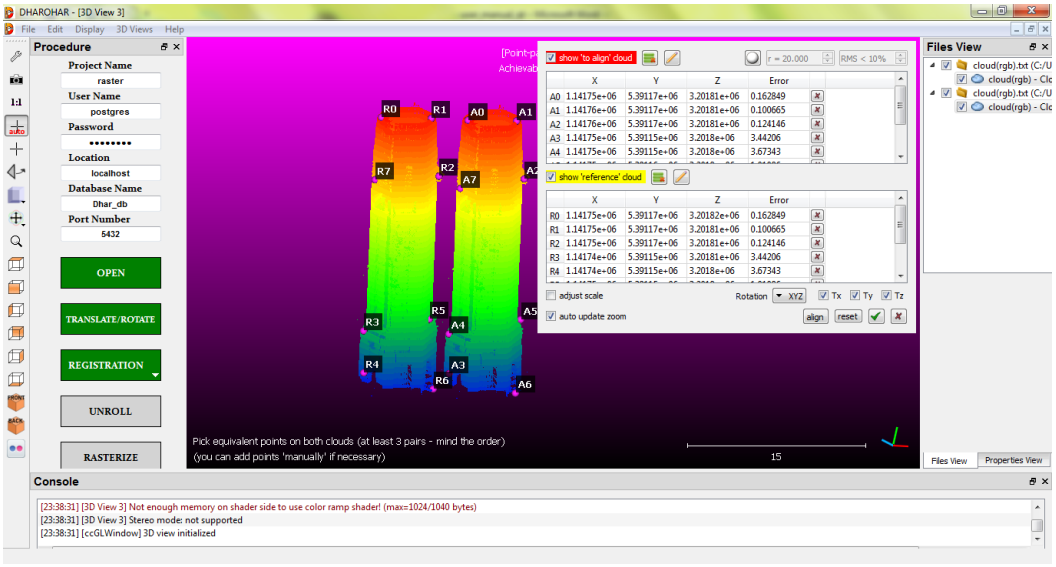


Fig 20: Points picked from both the cloud

- The points can also be picked manually by clicking on the manual icon.

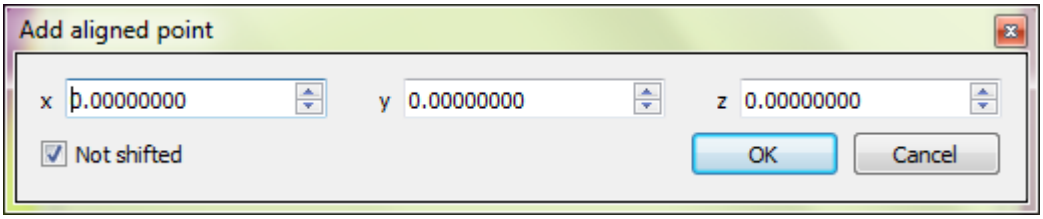


Fig 21: Add aligned point for registration

- Click on align button.

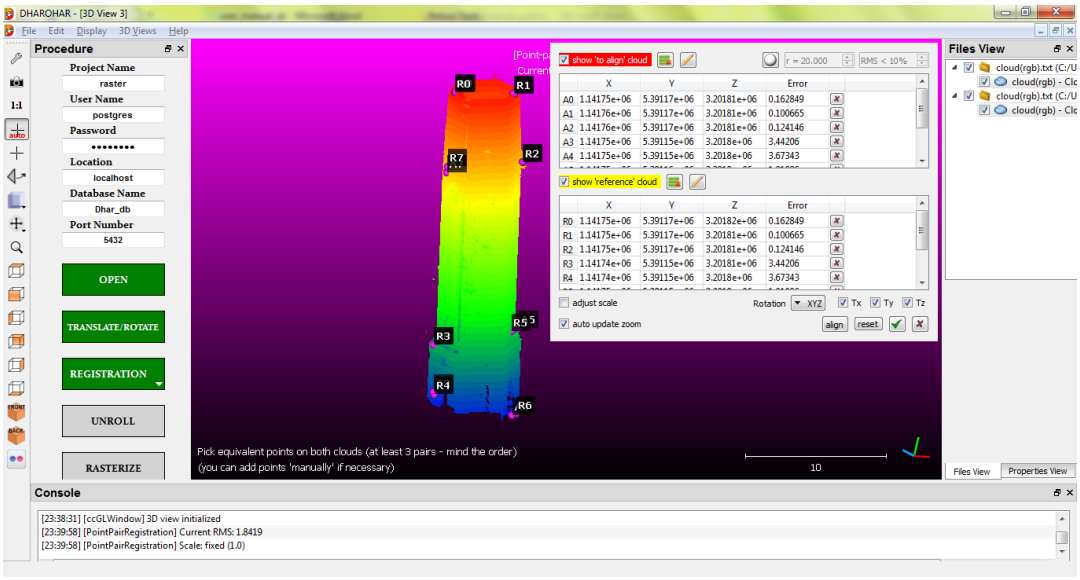


Fig 22: Registration applied

- Click *Tick* icon and the Align info will be displayed and click on *ok* button.

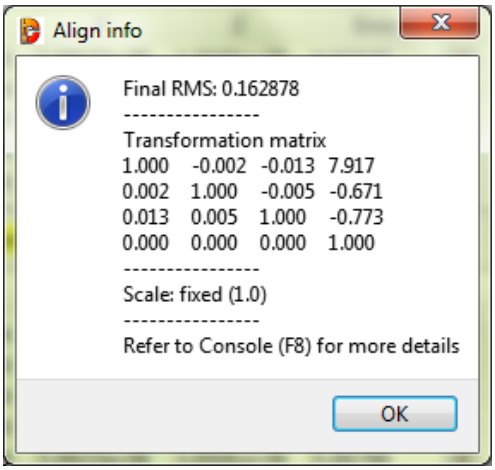


Fig 23: RMS output

Fine registration

This tool can automatically finely register two entities.

- Select the two point clouds both align and reference point cloud.
- And click on fine registration menu.

- This tool can automatically finely registers two entities.
- Main assumptions are:
  - both clouds are already roughly registered (see the other Alignment and Registration methods)
  - both clouds should represent the same object or at least have the same shape (at least on their overlapping parts)

Procedure

- Select the two clouds (or meshes) that you want to register and start this tool.
- We use the original ICP algorithm denominations here: first you have to choose which entity will be the 'Data' one (Registered, will eventually move) and which one will be the 'Model' one (Reference, won't move). You can the default role assignation by clicking on the 'swap' button. You'll see in the 3D view that the two entities colours are forced to yellow and red so as to correspond to the 'Model' (yellow) and 'Data' (red) colours.

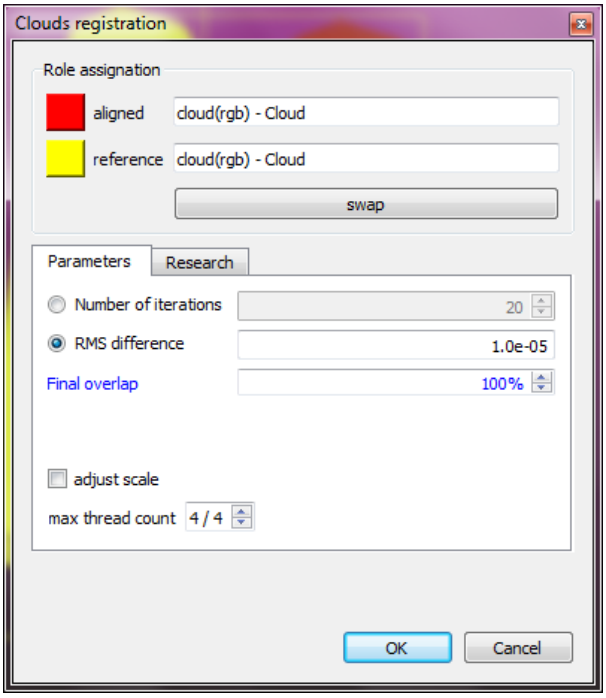


Fig 24: Fine registration

Main parameters:

Here are the most important parameters:

- **Number of iterations/RMS difference**: ICP is an iterative process. During this process, the registration error (slowly) decrease. We can tell CC to stop this process either after a maximum number of iterations, or as soon as the error (RMS) difference between two iterations becomes lower than a given threshold. The smaller this threshold is, the longer it will take to converge, but the finer the result should be (note: as CC work with 32 bits floating point values, a 1e-8 threshold is already near the computation accuracy limit and it shouldn't be necessary to go any lower).
- **Final overlap**: this is a new parameter for the version 2.6.1. It lets the user specify the actual portion of the data/registered cloud that would actually overlap the model/reference cloud if both clouds were registered. This let the user register entities with only a partial overlap (down to 10% or even less).
- **Adjust scale**: the modified-ICP algorithm we use is able to determine a potential difference in scaling. If your clouds have different scales (e.g. photogrammetry clouds) you can check this option so as to resolve the scaling as well.

Advanced parameters

You can optionally set additional research parameters (some of which are not yet validated, so that if you change them you might get unexpected results):

- **Random sampling limit**: to drastically increase computation speed on big clouds, we use an optimization scheme. It consists in randomly sub-sampling the data cloud at each iteration. This parameter is the maximum number of sub-sampled points. The default value (50000) is generally a good guess and its incidence on the result is not perceivable. However it may be insufficient for very large clouds. So if you doubt about the results, or if you want to refine the registration even more and you are not afraid of waiting a long time, don't hesitate to increase this value (to fully deactivate this optimization scheme, simply input a number greater than the data cloud size).
- **Rotation**: you can constrain the rotation around a given axis (X, Y or Z)
- **Translation**: you can constrain the translation along none, one or several dimensions (among X, Y and Z)
- **Enable farthest point removal**: this option is very interesting if the shapes of the two entities you are trying to register are quite different (either because the entities don't represent exactly the same object, or simply
- Because the noise on one entity is too high). This tells CC to remove at each iteration the points of the 'data' cloud that are too far from the 'model' cloud.
- **Use displayed model/data scalar field as weights**: this option should enable the user to use scalar values as weights (either on the data or the model cloud - it is not advised to use weights on both clouds at the same time)

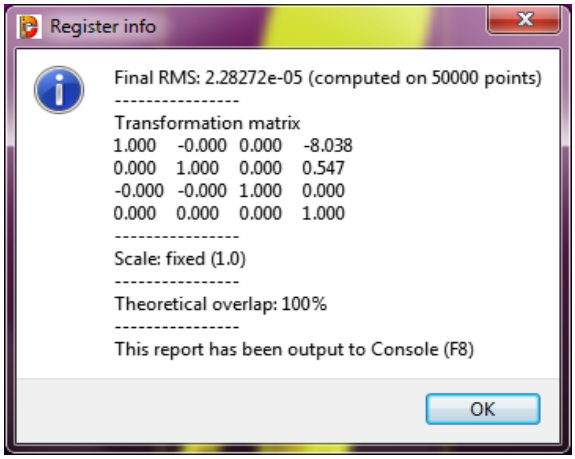


Fig 25: RMS Output for fine registration

(D) UNROLL

- Select the point cloud from the DB tree.
- Click on unroll button from the main menu.

This method 'unrolls' a point cloud from a cylindrical (or conical) shape onto a plane Cylinder

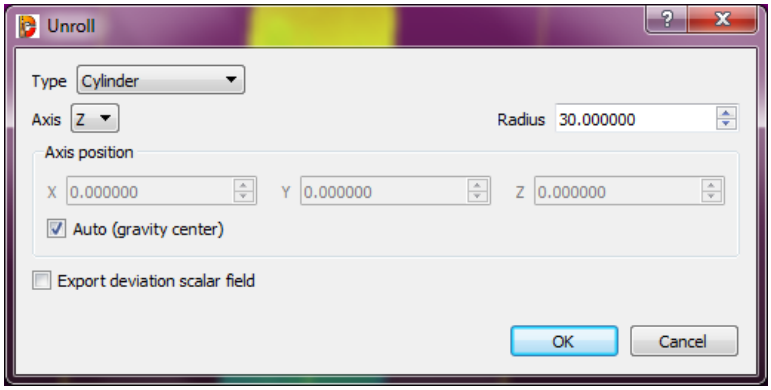


Fig 26: Unroll (Cylinder)

To unroll a cylindrical shape, the parameters are:

- axis of revolution (X, Y or Z)
- radius
- and optionally a point on the axis (*otherwise will use the cloud gravity center*)

Cone

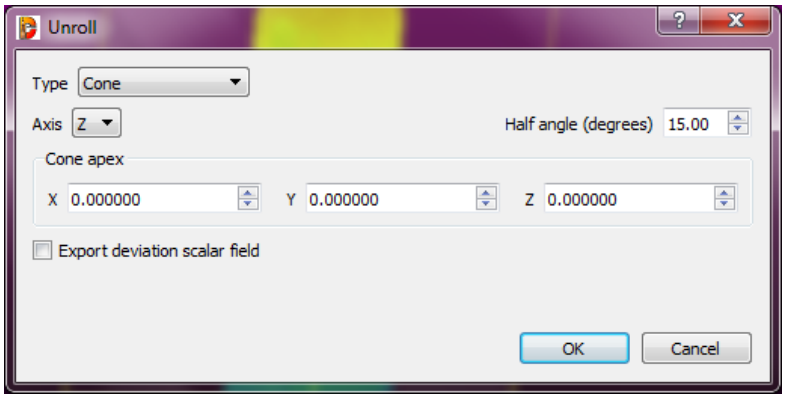


Fig 27: Unroll (Cone)

To unroll a conical shape, the parameters are:

- axis of revolution (X, Y or Z)
- half angle (this is the *aperture* angle at the cone apex - in degrees)
- the cone apex position

(E) RASTERIZE

- Select one point cloud from the db tree
- Click ok rasterize button from the main menu.

The main purpose of this tool is to 'rasterize' a point cloud (i.e. convert it to a 2.5D grid) and then export it as a new cloud or a raster image (geotiff) for instance.

A dedicated dialog with an embedded 3D view will appear. Some pieces of information about the selected cloud are displayed in the upper-left corner. The most important is the height range and the minimum and maximum height values of the cloud (*the height being considered along the projection dimension*).

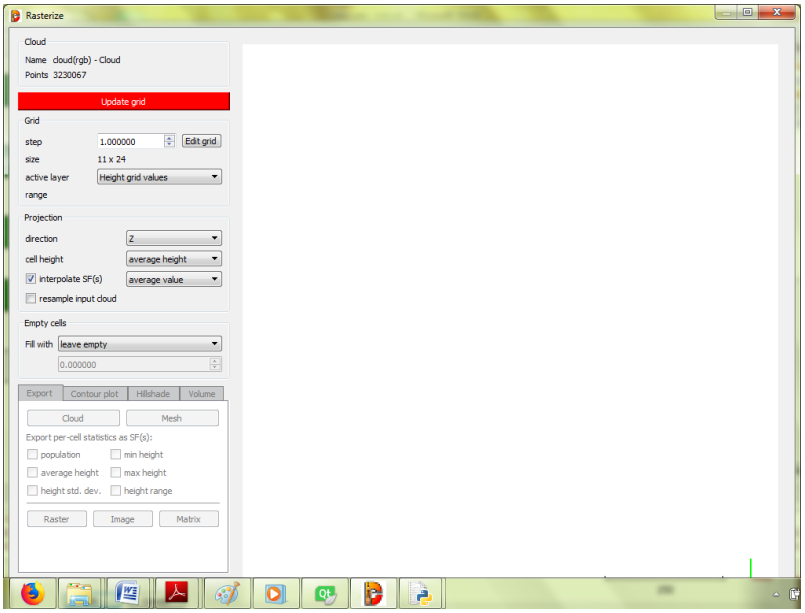


Fig 28: Rasterize window

Generating a raster grid

The first and mandatory step is to generate the raster grid.

The user must define the main (raster) grid generation parameters:

- the grid step size (Dharohar will update the resulting grid size below so that the user can check that the grid is neither too big nor too small before actually generating the grid)
- the projection direction (X, Y or Z - default: Z)

- how the 'height' of each cell grid will be computed:
    - minimum height of all points falling in this cell
    - average height of all points falling in this cell
    - maximum height of all points falling in this cell

Once the base parameters are properly set, the user must click on the 'Update grid' button to make Dharohar actually computed the grid and display it.
Each time a parameter is modified, the 'Update grid' button will appear in red. The user has to click on it to actually apply the changes.

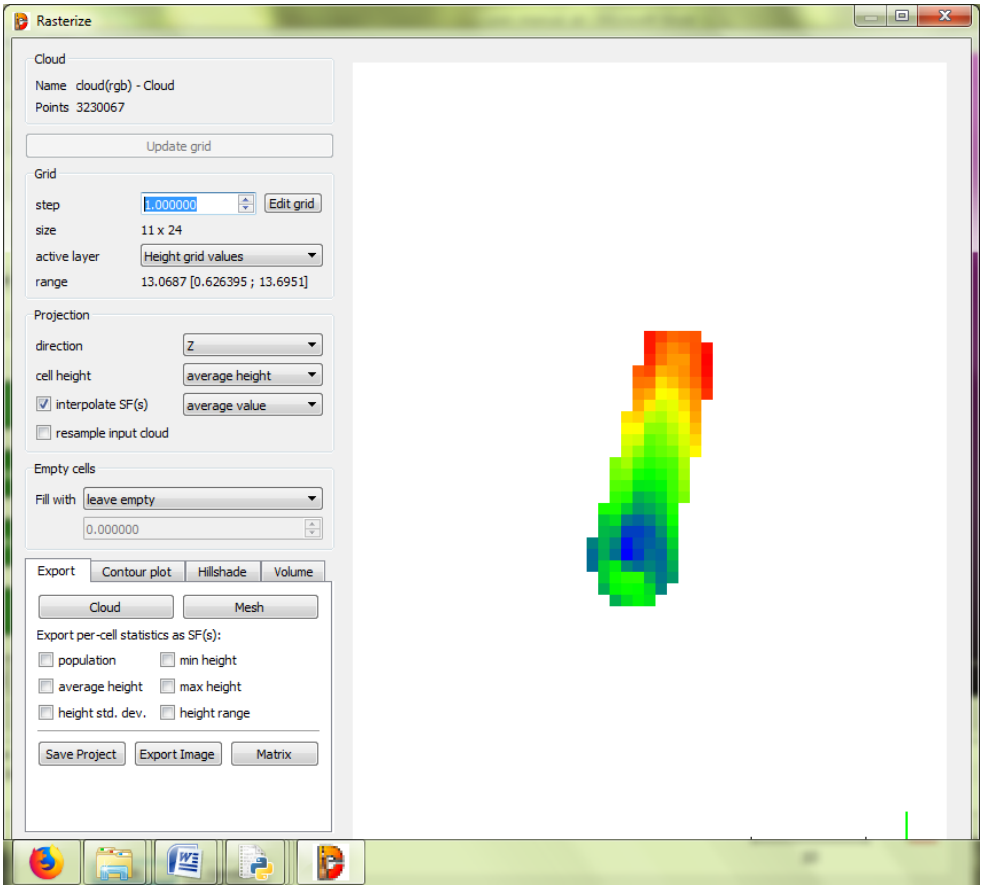


Fig 29: Rasterized image

Advanced parameters

Empty cells

If no point's fall inside a given cell, this cell will be considered as 'empty'. Empty cells are not visible when displayed by Dharohar in the 3D view. They generally have a dedicated 'NaN' or 'empty' value when exported to a raster file (depending on the format). It is possible to tell Dharohar to fill those cells in various ways:

- use the minimum height of the whole grid
- use the average height of the whole grid

- use the maximum height of the whole grid
- use a user specified value (should be input in the field below the 'Fill with' drop-down list)
- interpolate (see below)

Interpolating empty cells

The 'interpolate' option consists in a linear interpolation with the nearest non-empty neighboring cells. This can give very good results in the presence of small holes. However it can be less accurate on big holes. And above all it doesn't work outside the convex hull of the non-empty cells.

Grid position

The grid position (in the projection plane) can be finely defined by clicking on the 'Edit grid' button:

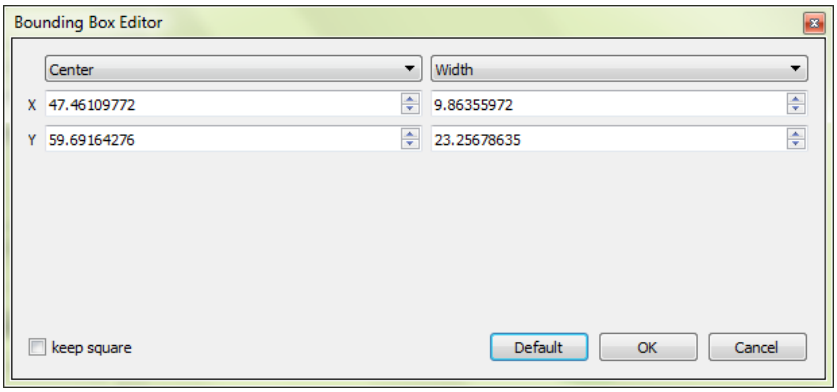


Fig 30: Bounding Box

Interpolate scalar fields

If the input cloud has one or several scalar fields, it is possible to 'interpolate' the scalar field values in each grid cell.

To do this the user has to check the 'interpolate SF(s)' checkbox and define how this interpolation should be conducted:

- by keeping the minimum SF value of all the points falling in this cell
- by keeping the average SF value of all the points falling in this cell
- by keeping the maximum SF value of all the points falling in this cell

Note: interpolated scalar fields can only be used when exporting the raster grid as a new cloud or as a raster file in a formats that supports real-valued layers.

Resample input cloud

This option tells Dharohar to keep in each grid cell the point which is the closest to the cell center (in 2D) instead of generating of using the cell center itself. This way it is possible to subsample the cloud in a semi-gridded pattern. If the grid is to be exported as a cloud, all the input cloud features (colors, normal, etc.) can be properly exported as well.

Cloud

- The grid can be exported as a new cloud (see the "Cloud" button in the 'Export' tab in the bottom-left corner).
- The grid is always exported as a 3D cloud (with the chosen 'height' as the 'Z' dimension). A 'height' scalar field is also generated by default.

Several additional scalar fields can be generated:

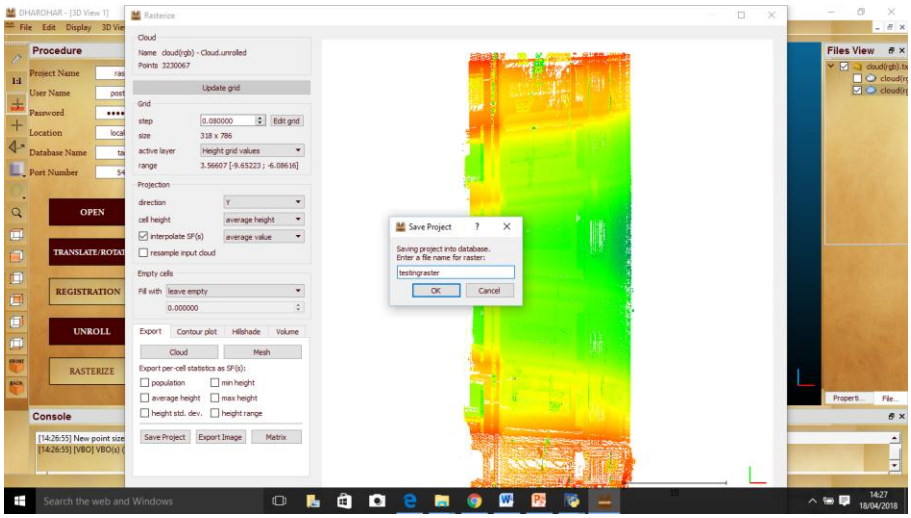
- 'population': number of input points falling in each cell
- 'min height': minimum height of the points falling in each cell
- 'average height': average height of the points falling in each cell (may be redundant with the default 'height' scalar field)
- 'max height': maximum height of the points falling in each cell (may be redundant with the default 'height' scalar field)
- 'average height': average height of the points falling in each cell (may be redundant with the default 'height' scalar field)
- 'height std. dev.': standard deviation of the height values of the points falling in each cell
- 'height range': range of the height values of the points falling in each cell

Export Image

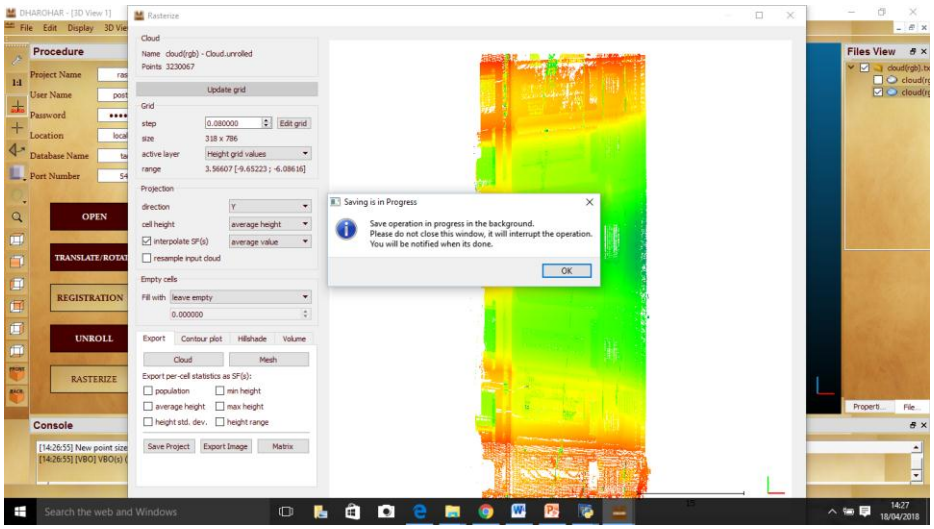
The grid can be exported as a simple image file. Use the 'Export Image' button.

Save Project

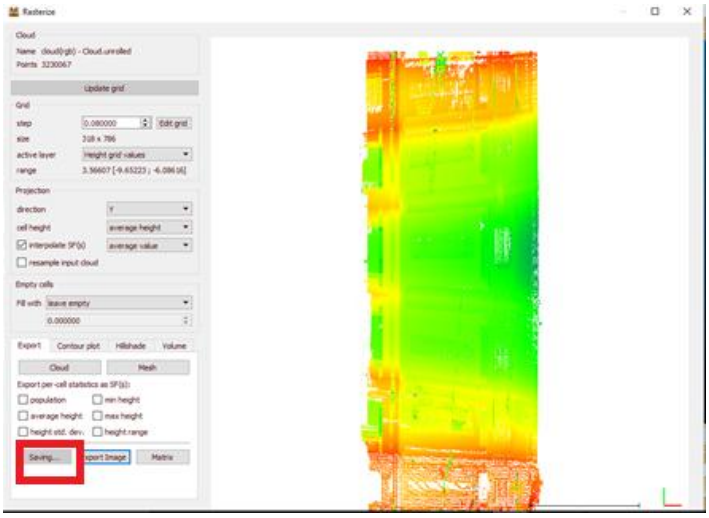
The Raster Image (tiff format) can be saved to the database. Use ‘Save Project’ button. On clicking Save Project a window appears as below:



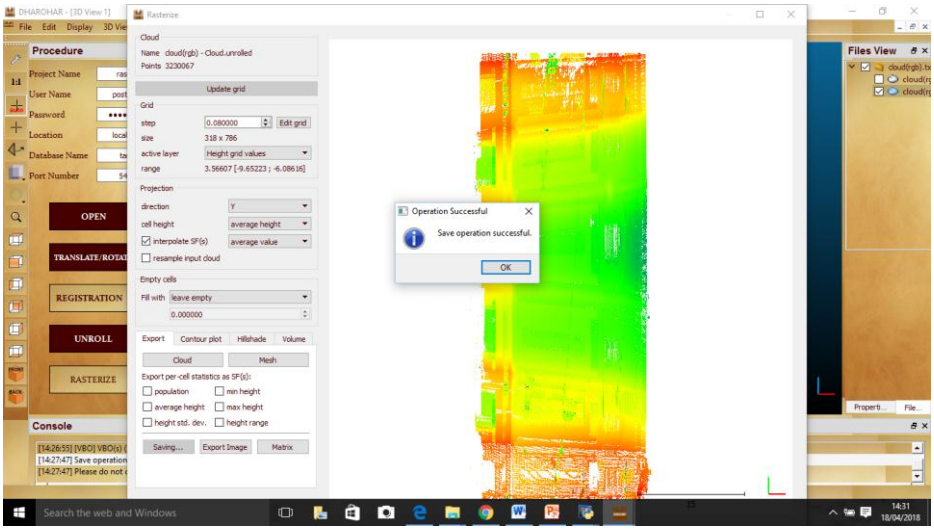
After giving the name of the project to be saved click on OK button. Then background processing will be taken place in order to save the project to the database.



Click on OK button on the message box that appears to start the process of saving.



Before project is saved need not close the window. A message box appears as follows after the project is saved.



ASCII matrix

The grid can be exported as an array/matrix of height values saved as an ASCII file (see the "Matrix (text file)" button in the 'Export' tab in the bottom-left corner).

This file should be easily imported in Excel for Matlab for instance. There's no file header. The number of rows is simply the number of lines in the file, and the number of columns corresponds to the number of values found on each line (should always be the same).

Counter plot generation:

The parameters for this sub-tool are all regrouped in the 'Contour plot' tab in the bottom-left part:

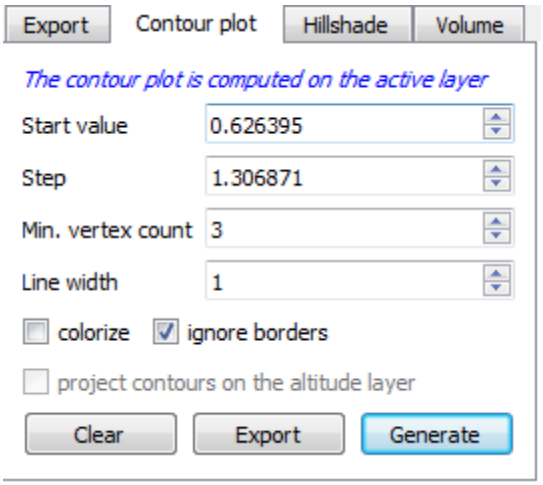


Fig 31: Contour plot

The user must specify:

- Start height: height of the first contour line
- Step: step between each contour line
- Min. vertex: minimum number of vertices per lines (used to remove the very small contours around trees, etc.)
- ignore borders: to remove the contour lines created on the grid (square) edges

The first time, and each time the parameters are changed or the grid is updated, the user must click on the 'Generate' button to generate the contour plot. Then a preview of the generated contour lines will be displayed over the raster grid:

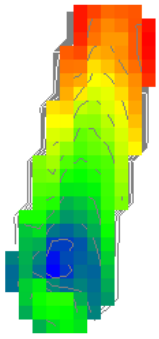


Fig 32: contour plot image

The user can pan and zoom the 3D view to view it better. The point size can also be modified in the standard way (+/- interactors appear when the mouse hovers the top-left part of the 3D view). The resulting contour lines can be removed (with the 'Clear' button) or exported as real polylines in the DB tree (with the 'Generate' button). All contour lines are exported in a single group (automatically named after the input cloud name and the 'Step' value).

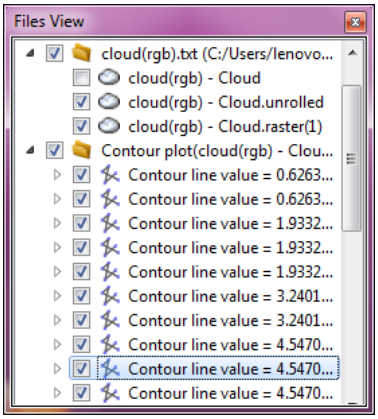


Fig 33: counter line value exported

Notes:

- The (group of) contour lines can be exported as a Shape file (to be imported in a GIS software for instance).
- If the user has forgotten to export the contour lines when closing the tool, Dharohar will issue a warning message and will ask for confirmation.

3.1.4 FILE MENU:

(A) **Open:** Refer 3.1.3 (A)
(B) **Open Recent:** It shows the recently opened files.
(C) Save:

- Click on File> save menu.
- This tool let the user save/export one or several entities to a file.
- Shortcut Ctrl+S.

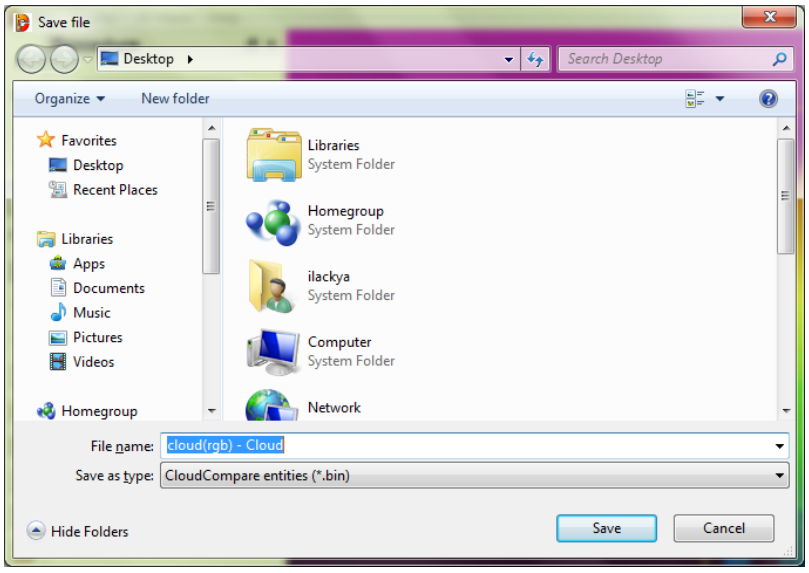


Fig 34: save point cloud to file

Select one or several entities then call this method.
Warnings:

- Be sure to select the right file type in the *Type* drop-down list.
- depending on the entities number and types, the available file types may change
- Some file types don't support non-ASCII characters (accents, non-occidental characters, etc.)

(D) Primitive Factor:

- Click on File>Primitive Factor Menu.
- The Primitive Factory lets you create primitive objects (planes, spheres, boxes, etc.).

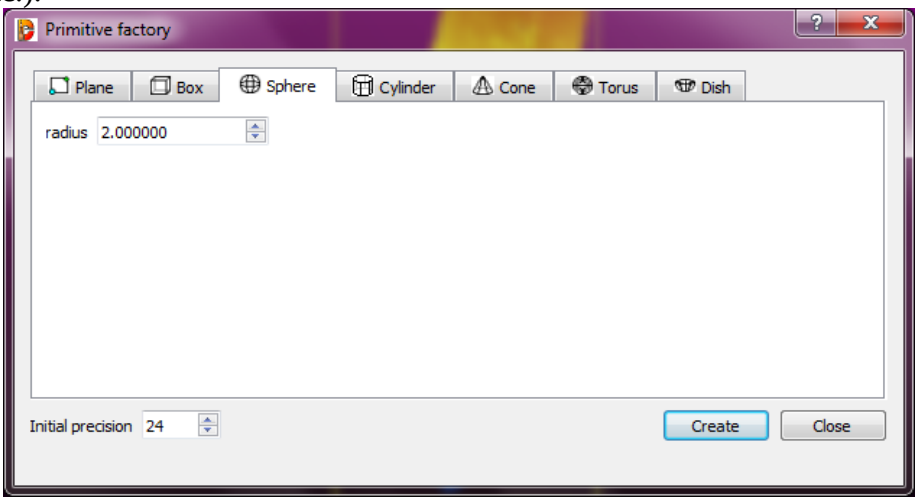


Fig 35: Primitive factor

You can choose the primitive type by selecting the right tab (Plane, Box, Sphere, Cylinder, Cone, Torus and Dish). For the selected primitive, you can edit its parameters (mainly dimensions) then create it by clicking on the 'Create' button. The process can be repeated several types (you can change the primitive type as well) until you close the factory with the 'Close' button.
Notes:

- Optionally you can also set the initial precision (i.e. the amount of triangles of the tessellated version of the primitive). However this parameter can be freely changed later in the primitive parameters

(E) Close all:

- Click on File>Close All Menu
- This tool let the user to close all the currently loaded entities

Note: Dharohar will ask the user to confirm this action (if entities are loaded).

(F) Quit:

- Click on File>Quit Menu
- This tool let the user to quit the application.
- Shortcut Alt+F4.

Note: Dharohar will ask the user to confirm this action (if entities are loaded).

3.1.5. DISPLAY MENU:

(A) **Full screen:**

- Click on 'Display > Full screen' menu.
- Alternatively you can use the *F11* shortcut.
- This method simply makes the main Dharohar application window full screen.
- Call this method again (via the menu or the *F11* shortcut) to restore its original state.

(B**) Refresh:**

- Click on 'Display > Refresh' menu.
- Alternatively you can use the *F5* shortcut.
- This method simply forces the active 3D view to refresh its content (*all OpenGL primitives are redrawn*).

(C) Toggle Centered Perspective

- This method is accessible via the icon in the left 'View' toolbar or the 'Display > Toggle Centered Perspective' menu.
- Alternatively you can use the *F3* shortcut.
- This method toggles the current projection of the active 3D view between the 'orthographic' and 'object-centered perspective' modes (see the ’Display modes’ section).

(D) Toggle Viewer Based Perspective

- This method is accessible via the icon in the left 'View' toolbar or the 'Display > Toggle Viewer-based Perspective' menu.
- Alternatively you can use the *F4* shortcut.
- This method toggles the current projection of the active 3D view between the 'orthographic' and 'viewer-based perspective' modes (see the ’Display modes’ section).

(E) Lock rotation about vert. axis

- Click on the 'Display > Lock rotation about vert. axis' menu to access this tool.
- Alternatively you can use the *L* shortcut.

- This method simply locks the camera rotation around the vertical (Z) axis (in the active 3D view).
- When activated a '[ROTATION LOCKED]' message will appear in the top part of the 3D view:

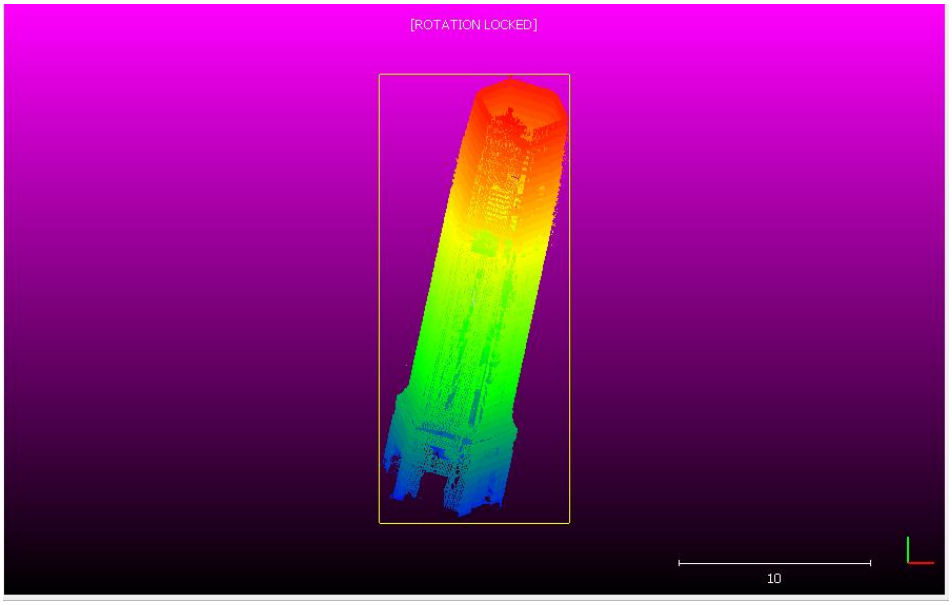


Fig 36: Rotation locked

- Simply call this method again (or use the *L* shortcut) to disable this mode.

(F) Enter bubble-view mode

- Click on the 'Display > Enter bubble-view mode' menu to access this tool.
- Alternatively you can use the *B* shortcut.
- This method enables the 'bubble-view' mode.

(G)Render to File

- Click on the 'Display > Render to file' menu to access this tool.
- This tool can 'render' the current 3D view as an image file (most of the standard file formats are supported). It can also apply a zoom so as to render the screen to a much higher resolution than the actual screen resolution.
- Make the 3D view that you want to render active then call this tool.
- A dialog will appear:

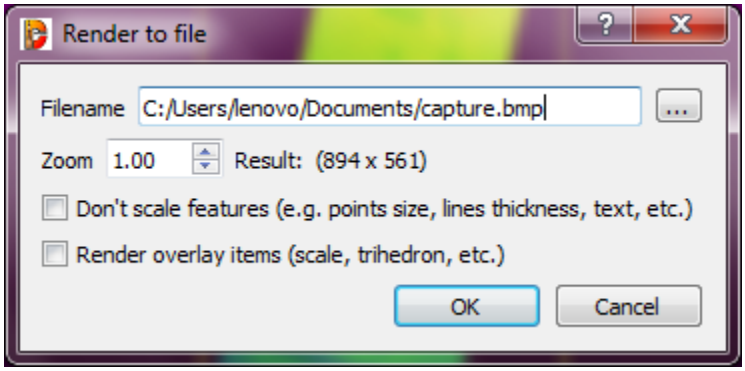


Fig 37: Render to File

You must first set the output image filename (you can use the '...' button to browse a particular file or folder on your computer). Most common image file formats are supported (bmp, jpg, png, etc.).

By default the output image will have the same resolution as the 3D view (i.e. the same size in pixels). With the 'zoom' factor, you can increase the rendered image resolution (the resulting size is displayed on the right). Dharohar will render the 3D view content *off-screen* in a potentially much larger buffer than you actual screen.

Warning: depending on your graphic card (and its driver) capabilities, the operation may fail if the output image size is too big. Most graphic cards / drivers have a limit of 64 M. pixels. Other options are:

- By default, is a zoom factor other than 1 is sued, Dharohar will 'scale' the displayed features (point size, lines thickness, etc.) when rendering the 3D view off-screen. The user can deactivate this behavior (this way, especially if the point cloud is very dense, a much finer rendering can be achieved).
- Moreover, by default the trihedron, the scale or other 2D overlay items are not rendered. The user can force Dharohar to render them.

(H) Display settings

- This method is accessible via the icon in the left 'View' toolbar or the 'Display > Display settings' menu.
- This method displays the 'Display settings' dialog.
- The options are regrouped in several tabs:

i. **Colors and materials**

The first tab regroups options related to the display of 3D entities.

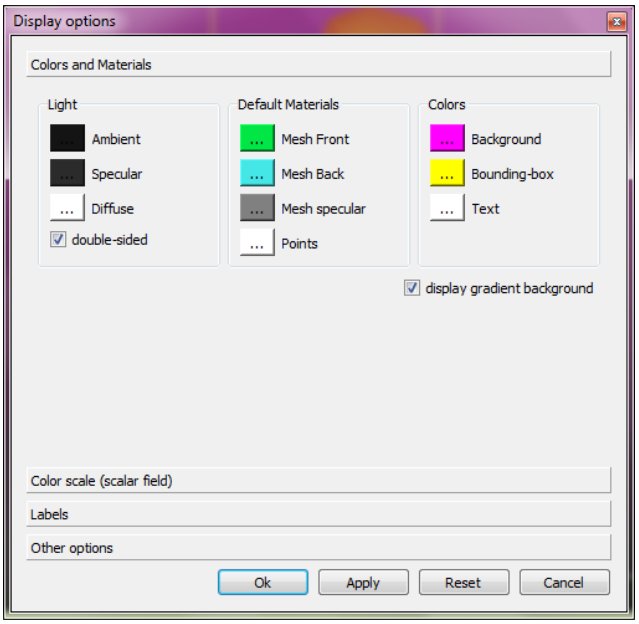


Fig 38: Display settings – colors and materials

One can set:

- the sun light components
- the default mesh material components (for triangular meshes without any material definition)
- the default point color (for clouds without any RGB color information)
- the colors of other display elements:
  - the 3D view background (if the *display gradient background* is checked, Dharohar will automatically create a gradient background with the 'Background' color and the inverse of the 'Points' color)
  - the (unselected) bounding-box color
  - the text color

ii. **Color sale**

The second tab regroups options related to the 'Color scale' display.

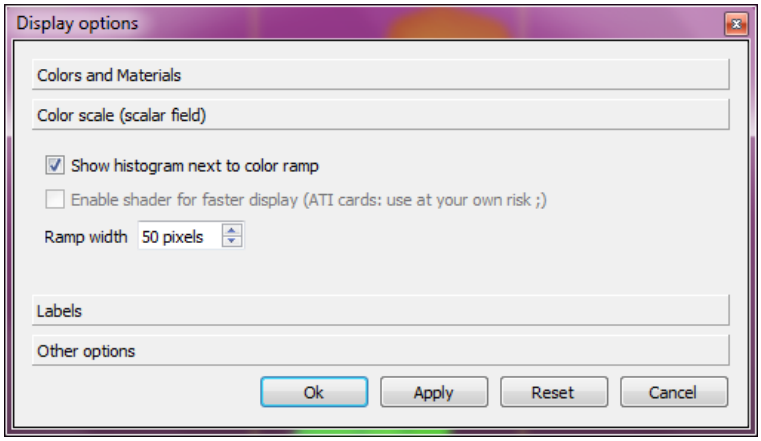


Fig 39: Display setting- Color scale

The user can set:

- whether the histogram should be displayed next to the color scale
- Whether the 'fast color scale display' shader should be enabled or not. This shader accelerates the display of dynamic colors for points when a scalar field is active. Depending on your graphic card it may not be possible to enable it. And due to some issues reported with ATI cards, it is deactivated by default on those cards (the user can still enable it however).
- the color ramp width

iii. Labels

The third tab regroups options regarding the display of labels.

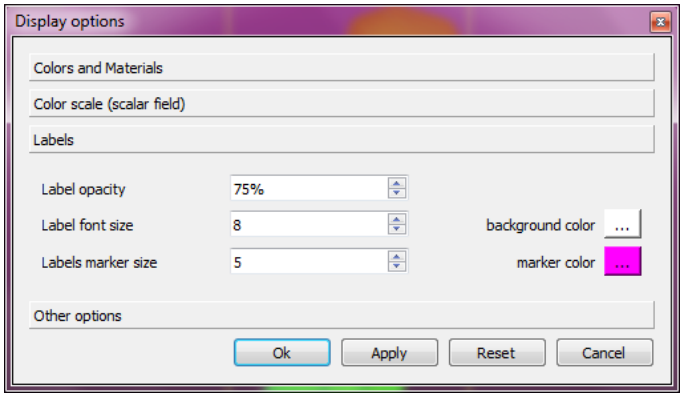


Fig 40: Display setting- Label

The user can set:

- the labels opacity
- the labels background color

- the labels font size (*different from the default text size since version 2.6.1*)
- the labels marker size (in 3D)
- the labels marker color (in 3D)

iv. **Other options**

The fourth tab regroups all the other parameters.

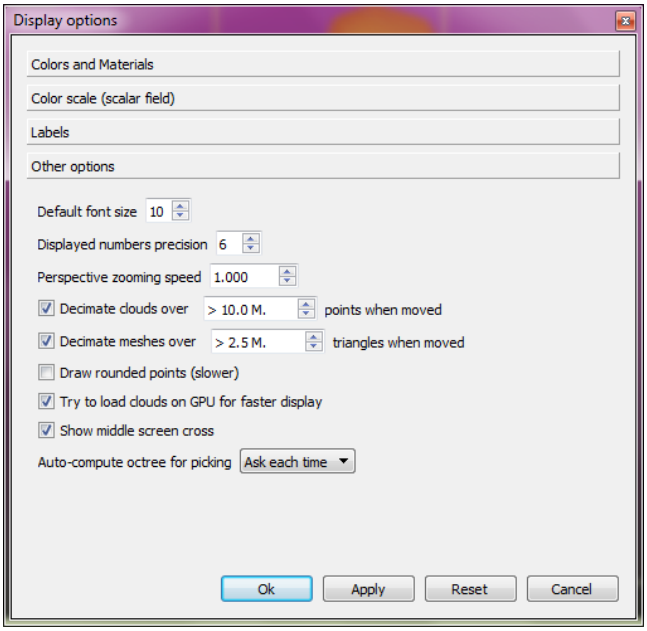


Fig 41: Display setting- Other Options

The user can set:

- the default font size
- the precision of displayed numbers (i.e. the number of digits)
- the zoom speed in perspective mode (since version 2.6.1)
- whether a cross should be displayed in the center of the 3D views (*middle screen cross*)
- whether big clouds (over 10 M. points) should be decimated when the view is rotated
- whether big meshes (over 2.5 M. triangles) should be decimated when the view is rotated
- Whether clouds should be loaded on the graphic card memory (thanks to VBOs [1]) - only activated on NVidia cards by default as crash were reported on ATI cards. The user can still enable this option at his own risks
- whether to use OpenGL picking mechanism for point picking (disabled by default as this feature is not accelerated and is generally slower than Dharohar's built-in algorithm)

(I) Camera settings

- This method is accessible via the icon in the left 'View' toolbar or the 'Display > Camera settings' menu.
- This method displays the 'Camera settings' dialog:

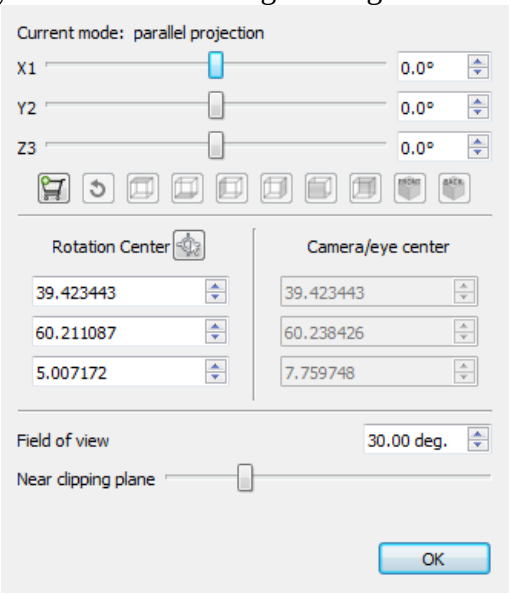


Fig 42: Camera settings

- **Main parameters**

  The user can change most of the parameters of the (OpenGL) camera of the active 3D view:

  - the camera orientation (with Euler angles: phi, theta and psi)
  - the scene rotation center (the icon lets the user pick a point in the 3D scene as new rotation center)
  - the camera/eye center
  - the field of view (only effective in Display modes perspective mode)
  - the near clipping plane (only effective in Display modes perspective mode)

Note: when changing a parameter the 3D view is directly updated. And conversely, when modifying the 3D view camera position or orientation while this dialog is opened the dialog parameters should be directly updated.

(J) Adjust zoom

- This tool is accessible via the 'Display > Adjust zoom' menu.

- This tool let the user define the current zoom, potentially in a very accurate way (e.g. in order to get a very specific dimension per pixel for instance).
- Make sure the targeted 3D view is active then call this tool.
- Dharohar will display a dedicated dialog:

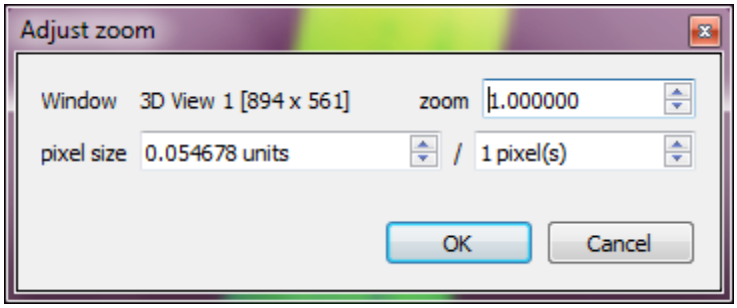


Fig 43: Adjust Zoom

- The zoom can be set in various ways:
  - either directly by setting the 'zoom' value
  - or by defining the dimension of one (or several) pixels. To do this set the 'pixel size' value to the right dimension, and optionally set the corresponding number of pixels (one by default). Dharohar will automatically update the corresponding 'zoom' value.

(K) Test Frame Rate

- This tool is accessible via the 'Display > Test frame rate' menu.
- This tool makes the active 3D view spin for a short time (~10 seconds) in order to estimate the average 'fps' (frame per second).
- The result is displayed in the Console.

(L) Console

The console is a dockable widget where all messages (standard information, warnings and errors) are traced. Some algorithms also output results in it.

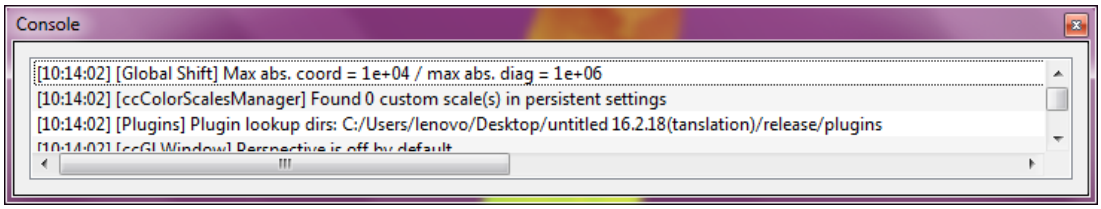
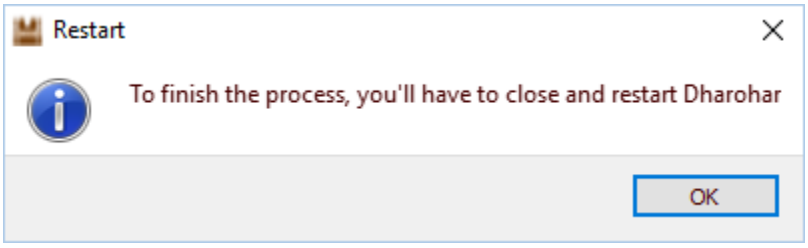


Fig 44: Console Window

(M) Reset all GUI elements

- This method is accessible via the 'Display > Reset all GUI elements' menu.
- By default Dharohar saves the current GUI configuration (position and visibility of the toolbars, etc.) when quitting. This tool can be used to restore the original configuration.

Note: Dharohar must be closed and restarted to apply the changes.



3.1.6 EDIT

(A) Cropping of Point Cloud

- Click on Edit > Crop Menu
- This tool is used to crop one or several clouds inside a 3D box.

Note: this tool creates new clouds (one for each input cloud - if the cloud is intersected by the 3D box)

Procedure

- Select one or several clouds before starting the crop. The standard 3D box editing dialog appears:

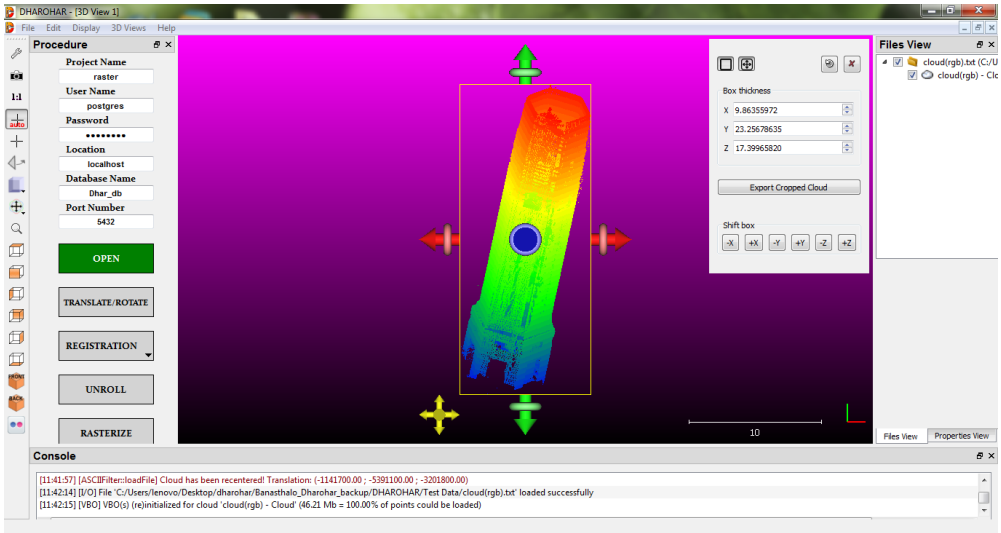


Fig 45: Cropping of point cloud

3D box editing dialog by default, the box is initialized to the bounding-box of all selected clouds (this default box can be restored anytime by clicking on the 'Default' button).

The user can define the cropping box in multiple ways:

- By defining the center and dimensions of the box [default]
- By defining the min corner and the dimensions of the box
- By defining the max corner and the dimensions of the box

Notes:

- For cubical boxes (the same dimension in all directions), you can check the 'keep square' checkbox.
- You can recall the previous box (if you call the tool several times) by clicking on the 'Last' button. The 'Last' button only appears if the tool has been called at least twice.

Eventually click on the 'OK' button to crop the input clouds and create the corresponding subsets (or click on the 'Cancel' button to cancel the process).

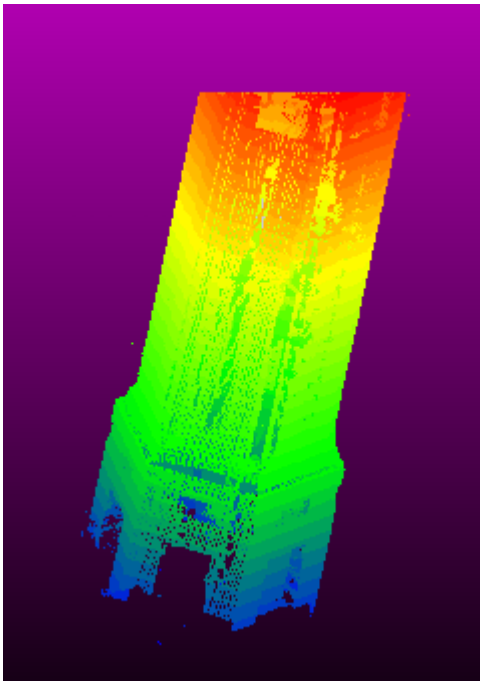


Fig 46: Cropped Point cloud

(B) Segmentation of Point Cloud

- Click on Edit > Segment Menu
- This tool allows the user to interactively segment the selected entities by defining a 2D polygon (or a rectangle) on the screen. This process can be repeated multiple times, changing the orientation of the entities each time, so as to properly segment the entities in 3D. Each time the user can decide

to keep the points (or triangles) falling inside or outside the polygon border.

Procedure

Select one or several entities and start the tool. A new tool bar will appear in the top-right corner of the 3D view.

Fig 47: Segmentation tools

Polygon edition mode

By default the tool starts in 'polygonal' editing mode. This means that you can start drawing the polygon right away:

- left click: create a new polygon vertex
- Right after the first vertex is created, you'll see that the first polygon edge will start to "follow" the mouse cursor. You have to define the position of the second vertex (left click) in order to 'fix' it. This process will start over with the next edge and so on.
- Right click: stop the polygon edition (warning: the currently 'floating' vertex won't be added to the polygon)

Rectangle edition mode

You can switch the 'Polygon edition' mode to 'Rectangle edition' mode by clicking on the down arrow next to the 'polygon' icon:

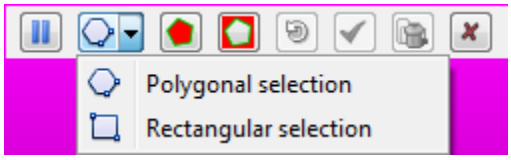


Fig 48: Rectangle edition modes

In 'Rectangle edition' mode you have to left click a first time to define the first corner of the rectangle then click a second time to define the opposite corner (alternatively you can keep the left mouse button pressed and the opposite corner will be created once you release the button).

Segmentation

Once the polygon/rectangle edition is finished, if the user clicks on the left button the edition process will start over (i.e. the current segmentation polygon will be discarded).

Otherwise the user has to choose whether to keep points inside or outside the polygon. Once done the other points will disappear (as well as the polygon). The tool will fall back in "paused" mode.

Paused mode

In paused mode, the mouse can be used to modify the entities orientation and position in the standard way.

The user has multiple choices:

- modify the current entities orientation and segment more points (click on the button to leave the 'paused' mode and define a new polygon/rectangle)
- reset the current selection
- validate the current segmentation and create two clouds: one with the selected points and one with the others
- validate the current segmentation and create only one cloud with the visible points - the other points will be deleted
- cancel the segmentation process(no modification will occur)

Export

The segmentation polygon can be exported as soon as it has been 'closed' (right-click) and BEFORE segmenting the points (inside or outside).

(C) Merging of Point Cloud

- Click on Edit > Merge Menu
- This tool merges two or more entities.
- It does Merging of point clouds

Warning: when merging clouds, the original clouds will be deleted (you may have to save or clone them first).

3.1.7 3D Views menu

(A) New

- Create a new 3D view.
- Shortcut: *CTRL + F3*

Note: entities can be moved to 3D views by editing their ‘with the ‘Current Display’ property.

(B) Close

- Closes the active 3D view.
- Shortcut: *CTRL + F4*

(C) Close All

- Closes all 3D views.

(D) Tile

- Share the display space between all the 3D views (as ‘tiles’).

(E) Cascade

- Rearrange all the 3D views in a ‘cascade’ way.

(F) Next

- Activate the ‘next’ 3D view (order of creation).

(G) Previous

- Activate the ‘previous’ 3D view (order of creation).

3.1.8 Help menu

(A) Help

- This method is accessible via the 'Help > Help' menu.
- Alternatively this method can be called with the *F1* shortcut.
- In a perfect world this method should display the user manual of dharohar.

(B) About…

- This method is accessible via the 'Help > About...' menu.
- This method displays a dialog with the current version information:

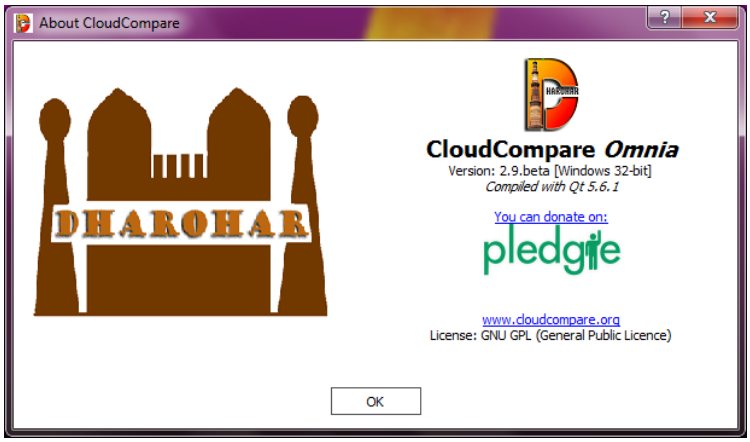


Fig 49: Display setting- Color scale

3.1.9 3D view toolbar

Fig 50: 3D view toolbar

Note: this toolbar is situated on the left side by default.

3.2 LOAD IMAGE LIST FROM DATABASE:

- The list images present in the database can be view by clicking on load/ refresh image list from database.
- The image list will be displayed in the list view with the Image ID and the File Name.

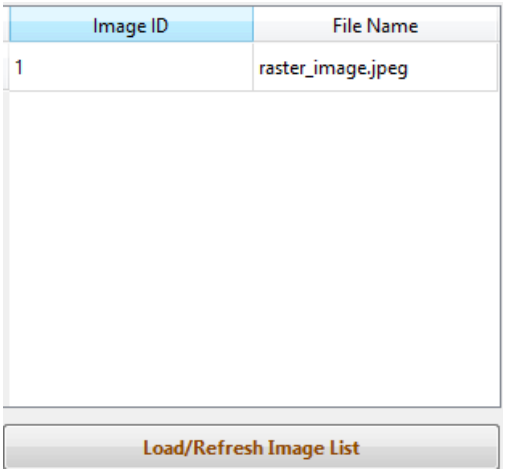


Fig 51: List of raster image from database

3.3 IMAGE ANALYSIS:

- Click on *image analysis* tab to access this tool.
- This tool is an image processing technique for finding the boundaries of object with raster images, detecting the corners of object and also converting RGB to HIS and vice versa.
- It works by detecting the discontinuity in brightness.
- The window appears as follows:

Fig 52: Image Analysis

Procedure:

- Image analysis techniques can be performed by loading the image either from the database or from the file.
- For loading image from the database :
  - Enter the *image id* value.
  - Click on *load Image from Database* button to load the image.
- Loading image from the files:
  - To load images from files click on *load image* button.

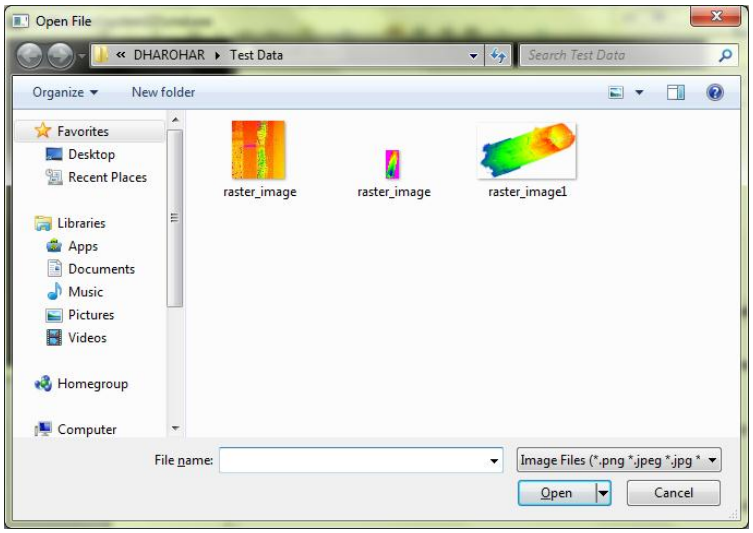


Fig 53: Load raster image from file

  - Select the required raster image from the files and click on *open* button.
- The Loaded Image can be viewed by clicking on *view loaded image* button.
  - After loading the raster image the image analysis techniques can be performed.
  - Image analysis includes :
    - Edge detection
    - Corner detection
    - RBG to HIS
    - HIS to RGB

3.3.**1 EDGE DETECTION**:

Edge detection includes a variety of mathematical methods that aims at identifying points on a digital image at which the image brightness changes sharply or, more formally, has discontinuities. The points at which image brightness changes sharply are typically organised into a set of curved line segments termed edges. The same problem of finding discontinuities in one dimensional signals is known as step detection and the problem of finding discontinuities over time is called change detection.

(A) LAPLACIAN:

- Check the *laplacian* button and click on *process* button.

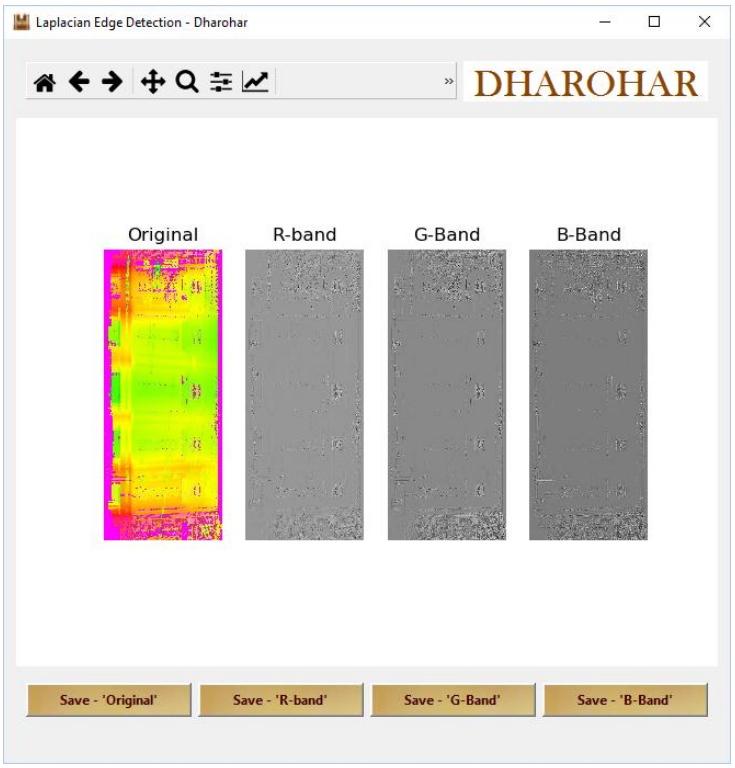


Fig 54: Laplacian output

(B) SOBEL:

- Check the *Sobel* button and click on *process* button.

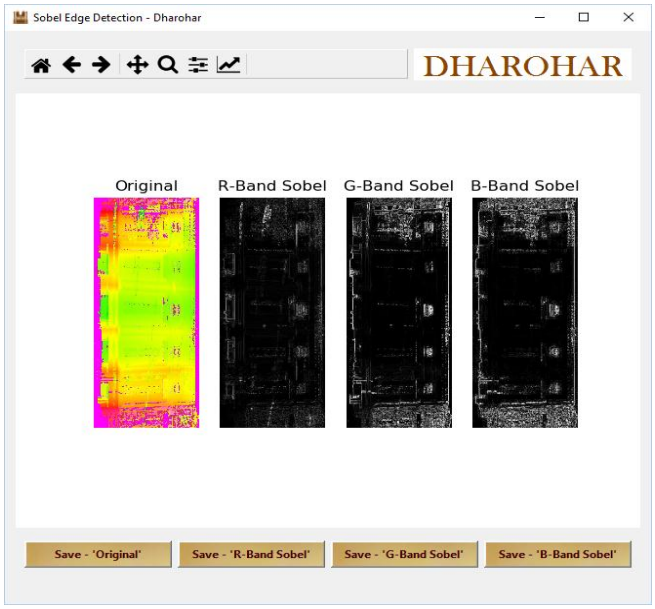


Fig 55: Sobel output

(C) PERWITT:

- Check the *perwitt* button and click on *process* button.

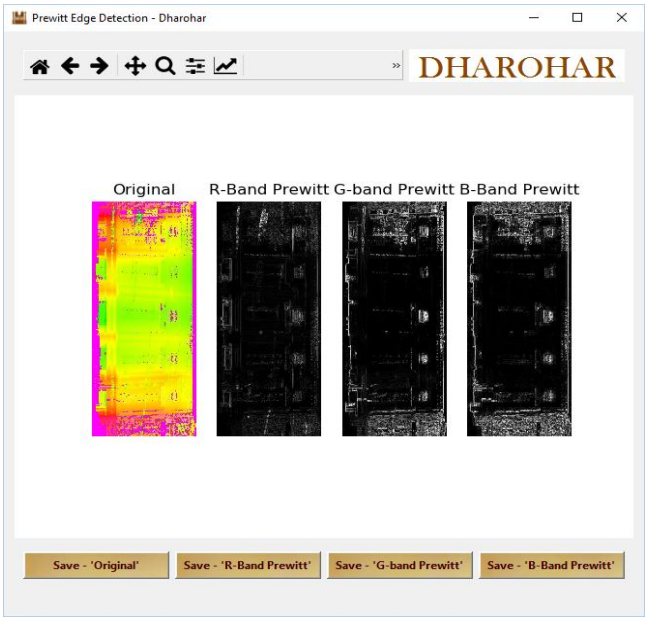


Fig 55: Perwitt output

(D) SCHARR:

- Check the *scharr* button and click on *process* button.

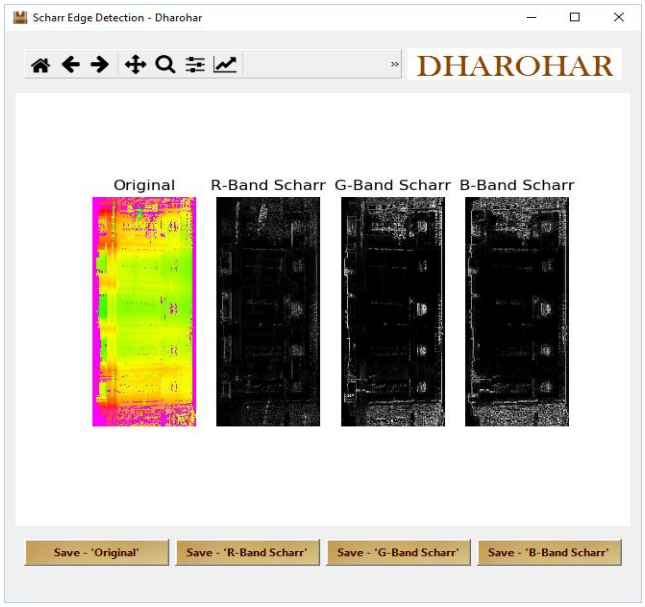


Fig 56: scharr output

(E) CANNY:

- Check the *canny* button and click on *process* button.

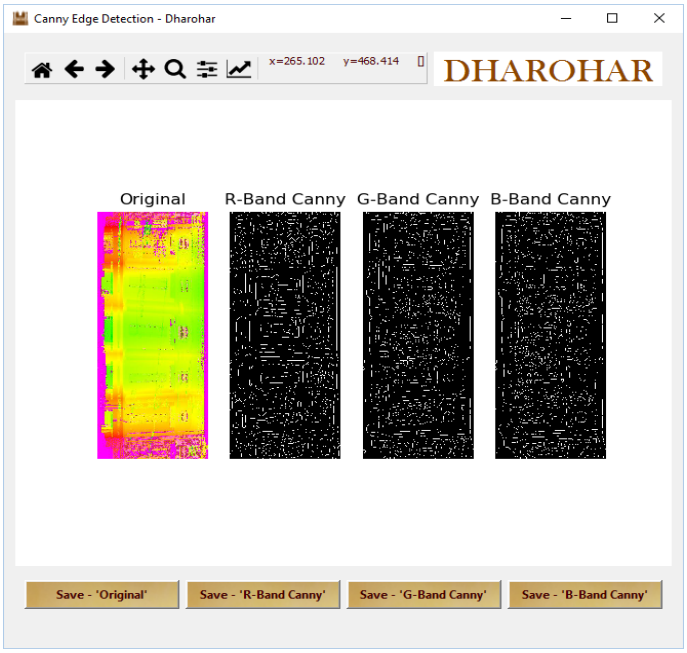


Fig 57: Canny output

3.3.2 **CORNER DETECTION**:

Corner extraction is an approach used within computer vision systems to extract certain kinds of features and infer the contents of an image. Corner detection is frequently used in motion detection, image registration, video tracking, image mosaicing, panorama stitching, 3D modelling and object recognition.

- Check the *corner detection* button and click on *process* button.
- A window appears with the corner detected in the raster image opened.

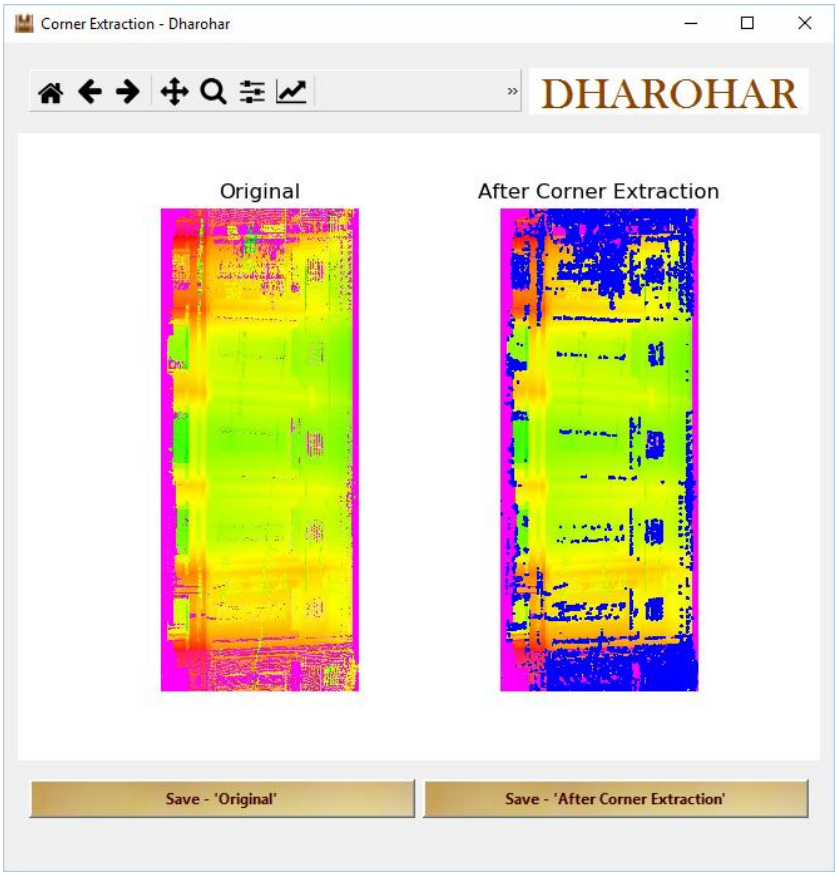


Fig 58: Corner detected output

3.3.3 RGB TO IHS:

In this feature, it simply generates the HIS (intensity, hue and saturation) image of the input RGB (red, green, blue) image.

- Check the *RGB TO HIS* button and click on *process* button.

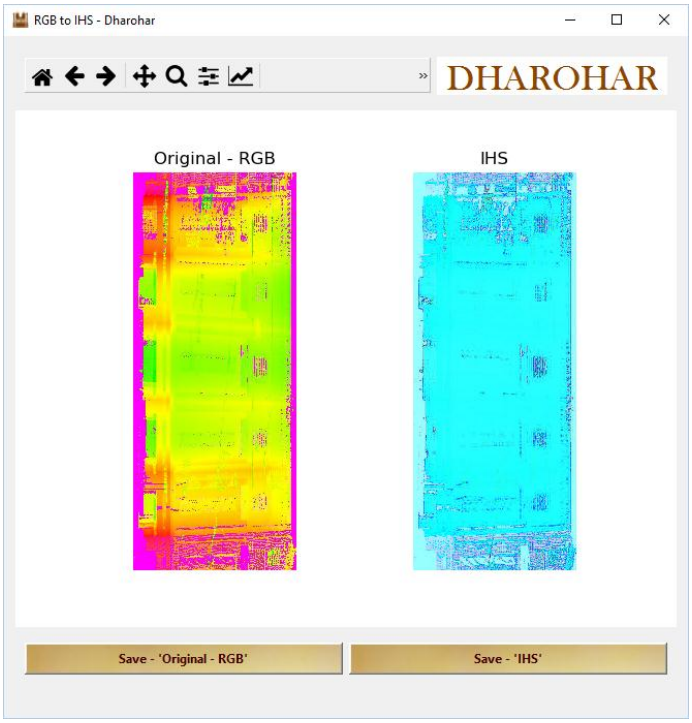


Fig 59: RGB to IHS converted Image

3.3.4 IHS TO RGB:

In this feature, it simply generates RGB (red, green, blue) the image of the input image HIS (intensity, hue and saturation).

- Check the *HIS to RGB button* and click on *process* button.

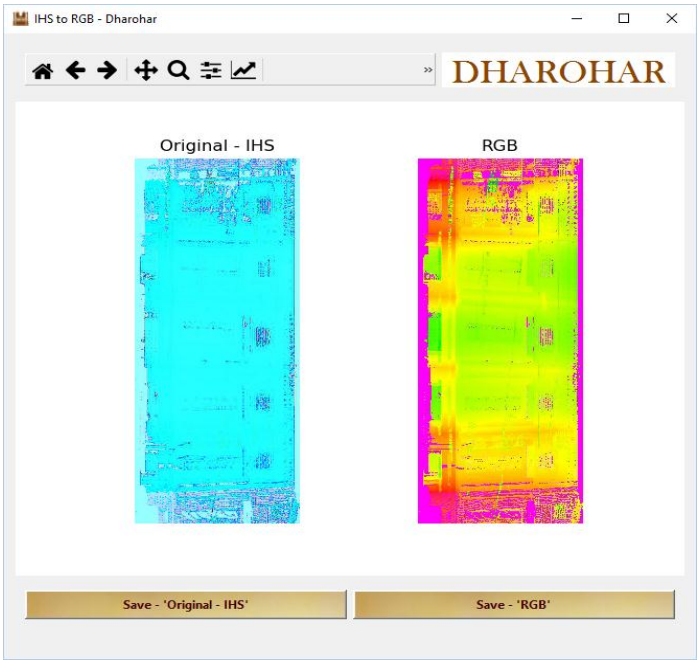


Fig 60: IHS to RGB converted Image

3.4 CHANGE DETECTION:

- Click on *change detection* button to access this tool.
- In statistical analysis, change detection or change point detection tries to identify times when the probability distribution of a stochastic process or time series changes. In general the problem concerns both detecting whether or not a change has occurred, or whether several changes might have occurred, and identifying the times of any such changes.

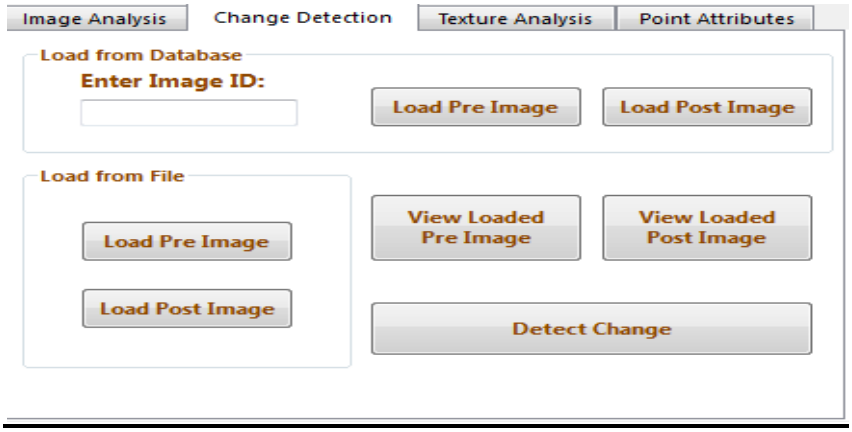


Fig 61: Change detection

- After loading the pre-image and post image click on process button.
- A window appears with the pre-image, post-image and the changes detected image as follows:

Fig 62: Change detected Image

3.5 TEXTURE ANALYSIS:

- Click on *texture analysis* button to access this tool.
- A statistical method of examining texture that considers the spatial relationship of pixels is the gray-level co-occurrence matrix (GLCM), also known as the gray-level spatial dependence matrix. The GLCM functions characterize the texture of an image by calculating how often pairs of pixel with specific values and in a specified spatial relationship occur in an image, creating a GLCM, and then extracting statistical measures from this matrix.
- After you create the GLCMs, using gray co-matrix, you can derive several statistics from them using gray coprops. These statistics provide information about the texture of an image.
- A new window appears as follows:

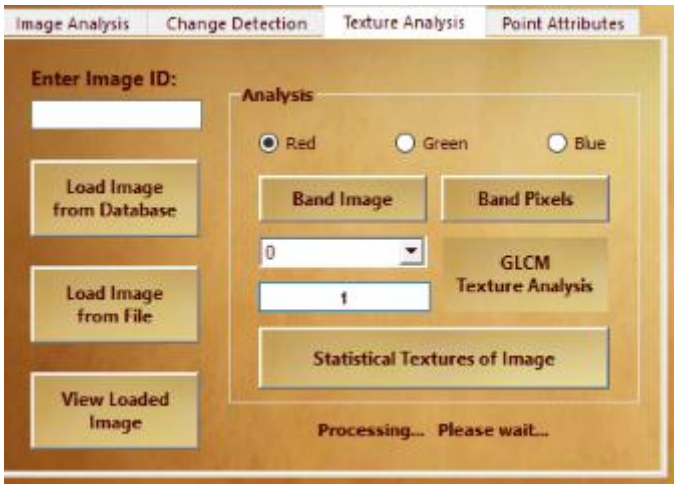
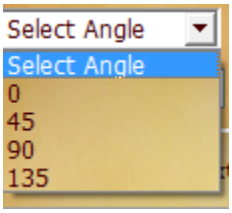


Fig 63: Texture analysis

- After loading the raster image we can perform the texture analysis technique

- Select either red, green or blue and click on Band Image button and also we can give different distance and angle value for GLCM Texture Analysis.



- . Output appears as follows:

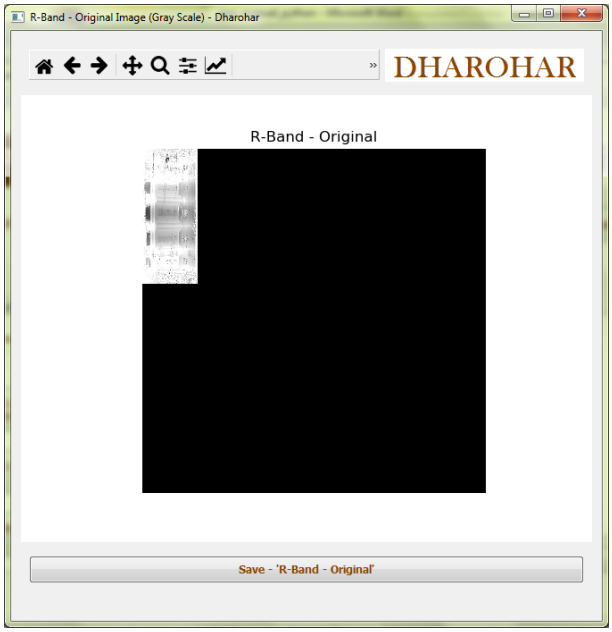


Fig 64: R-Band Original image

- Select either Red, Blue or Green and click on Band Pixel button. Output appears as follows:

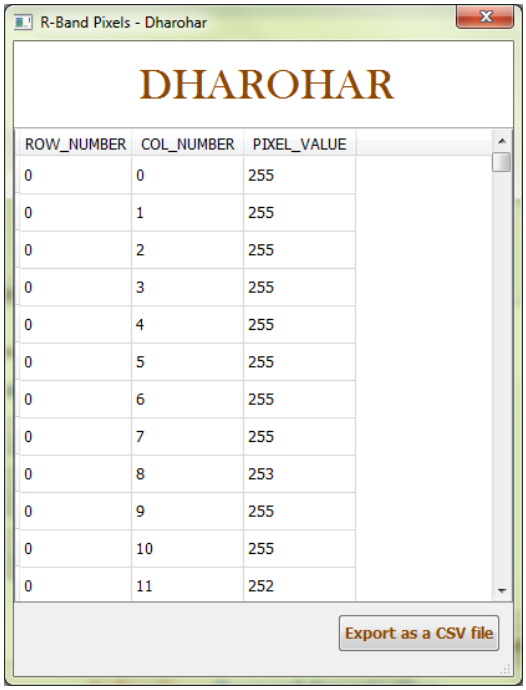


Fig 65: R-Band Pixels

- Select Red, Blue or Green and click on Statistical texture on image. The statistical texture on the image appears as follows:

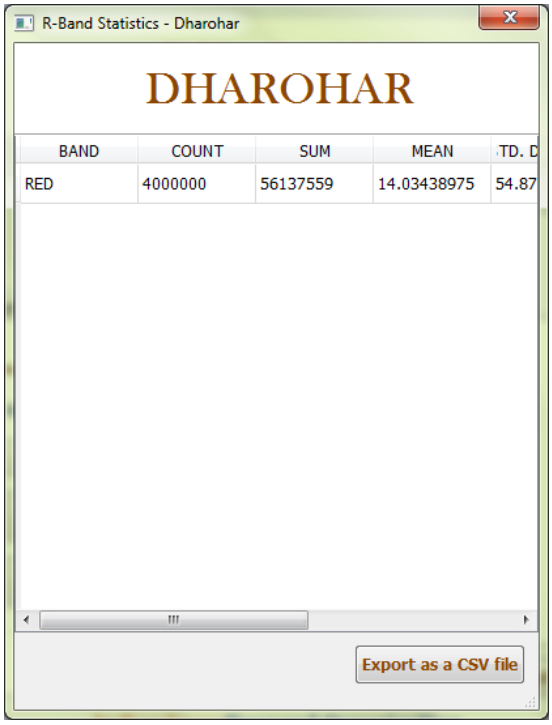


Fig 66: R-Band statistics

- Select Red, Blue or Green and click on GLCM texture analysis and also give distance and angle. The output image appears as follows:

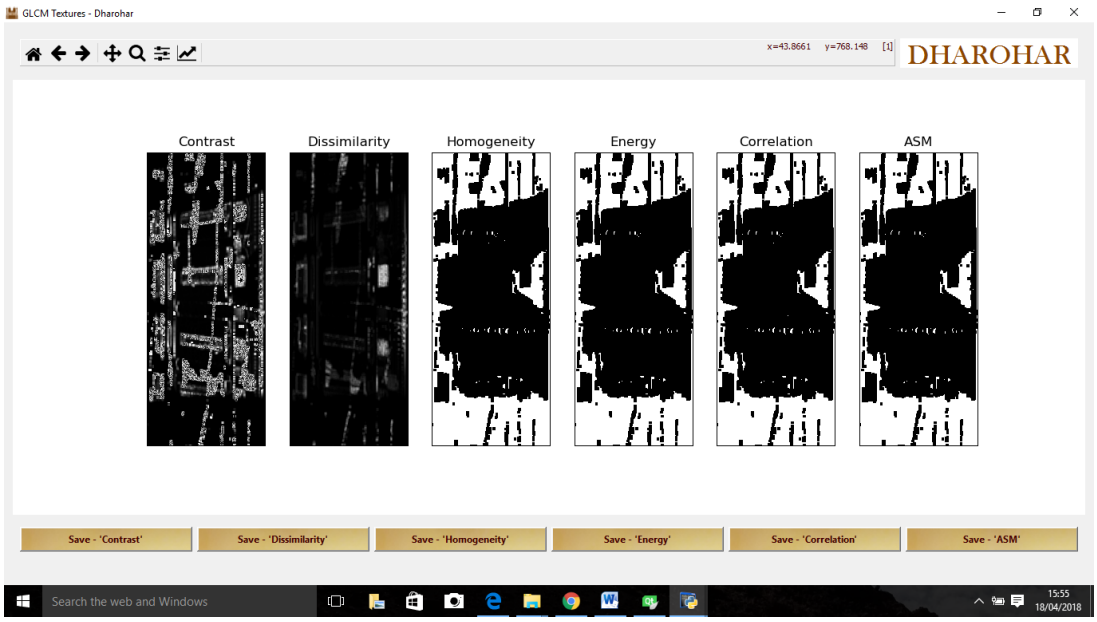


Fig 67: GLCM texture analysis output

3.6 POINT ATTRIBUTES:

- Click on *Point Attribute* Tab.
- Enter the Image ID to be loaded from the database.
- Select either 3D point attributes or Radius, Depth, Theta.
- Select the required fields to be visualized.
- And click on *Visualize* Button.

The visualization of attributes appears as follows:

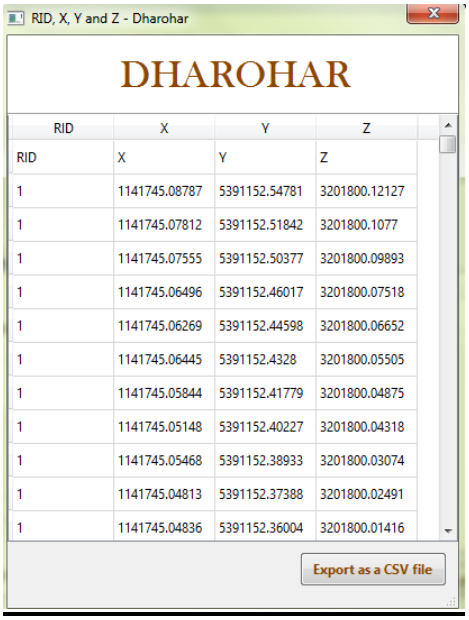


Fig 68 : RID, X,Y,Z extracted from database