



ShapePopulationViewer

User Tutorial V1.0.1

Alexis Girault, Francois Budin, Beatriz Paniagua, Martin Styner
Neuro Image Research and Analysis Laboratories
University of North Carolina at Chapel Hill



ShapePopulationViewer

Installation



Installation

You can either :

- Download the latest binary package on NITRC :

<https://www.nitrc.org/projects/shapepopviewer>

- Download the source code on GitHub :

<https://github.com/NIRALUser/ShapePopulationViewer>

To get the source from a Linux Shell :

```
$ git clone https://github.com/NIRALUser/ShapePopulationViewer
```

```
$ git checkout v1.0.1
```



Via GitHub

If you downloaded the sources on Github, you will have to build the project using **Cmake v2.8.3** or higher.

You can then specify which kind of build you want to make using the boolean variable `ShapePopulationViewer_SUPERBUILD`.

- **Qt 4** is required (*tested with Qt4.7.4.*)
- If you don't want to use the superbuild option, you will need:
 - **VTK 5** built with QT 4 (*tested with VTK5.10.1.*)
 - **SlicerExecutionModel** built with ITK 4 (*tested with ITK4.1.*)



ShapePopulationViewer

GUI : Basic Usage



GUI

Menu Bar → File Options

Display Area →

Information box →

Tool box →



Loading mesh(es)

There are multiple ways to load surfaces using the *File* menu:

- Loading vtk file(s) with “Open Files”
 - > select the files you want to load
- Loading an entire directory with “Open Directory”
 - > all the vtk files in the selected directory will be loaded
- Loading files from a Comma Separated Value file with “Load CSV”
 - > your file will be open in a spreadsheet to allow you to select the files you want to load



Add and delete meshes

ADDING

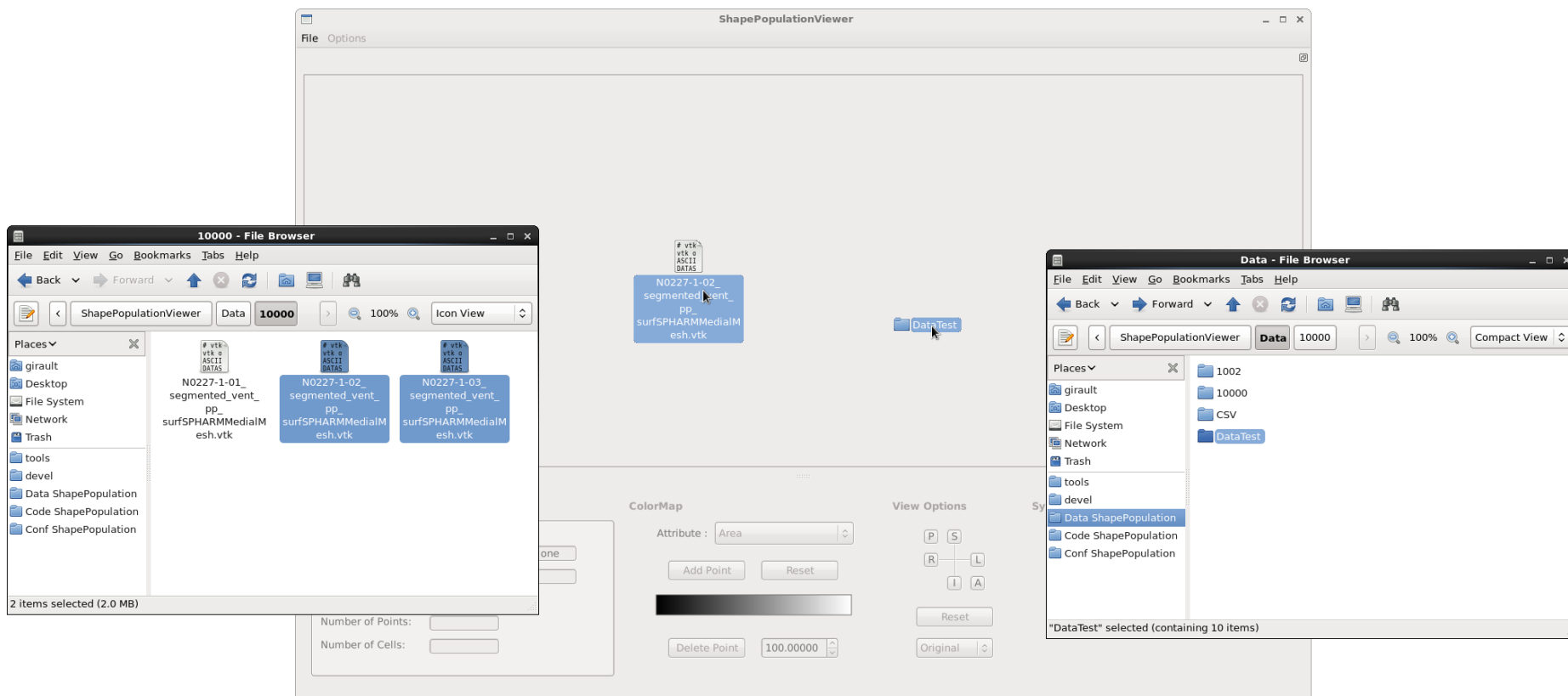
Once meshes are displayed in the Display Area, “Open” become “Add” : loading new files will add the meshes to the current display window.

DELETING

You can delete the selected items or all the items using the options in the *File* menu : “Delete Selection” and “Delete All”.



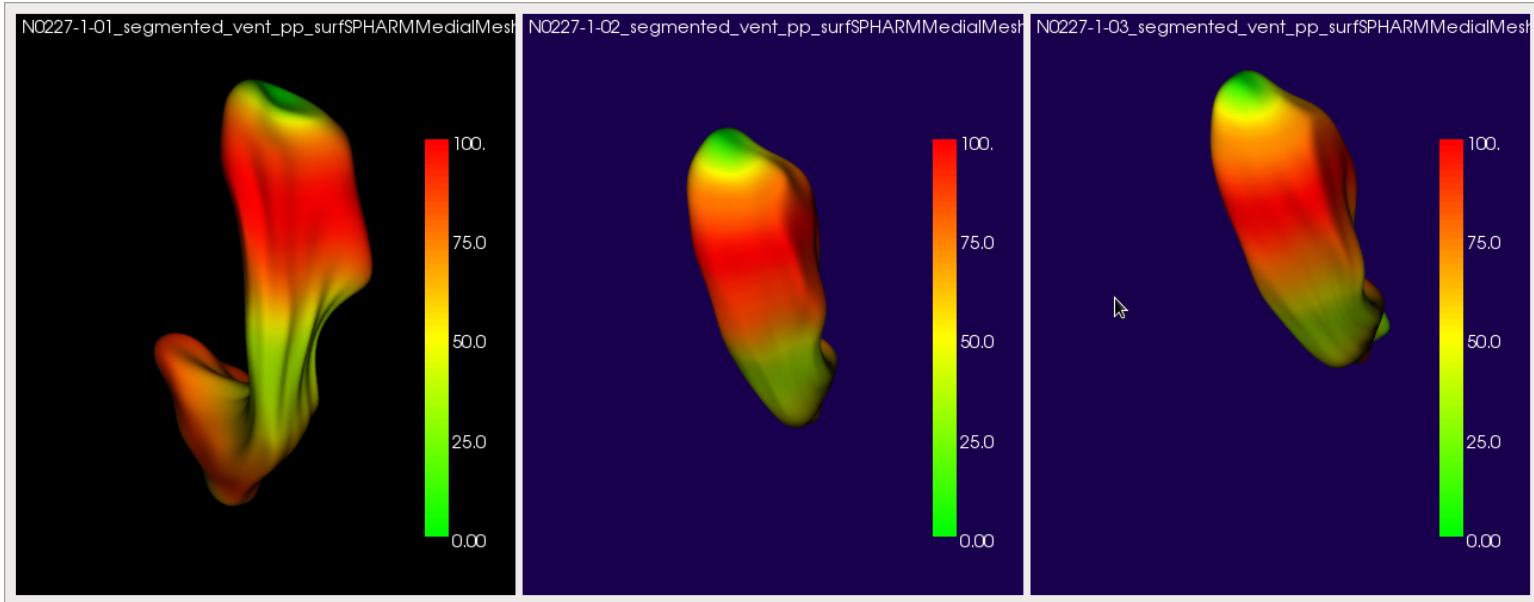
Drag & Drop meshes



You can also load/add files (.vtk files, directories or CSV files) by dropping them over the Display Area.



Interacting with surfaces



In the Display Area, you can interact with multiple surfaces at the same time by selecting their respective windows and then interacting (click & drag, roll...) with only one of them :

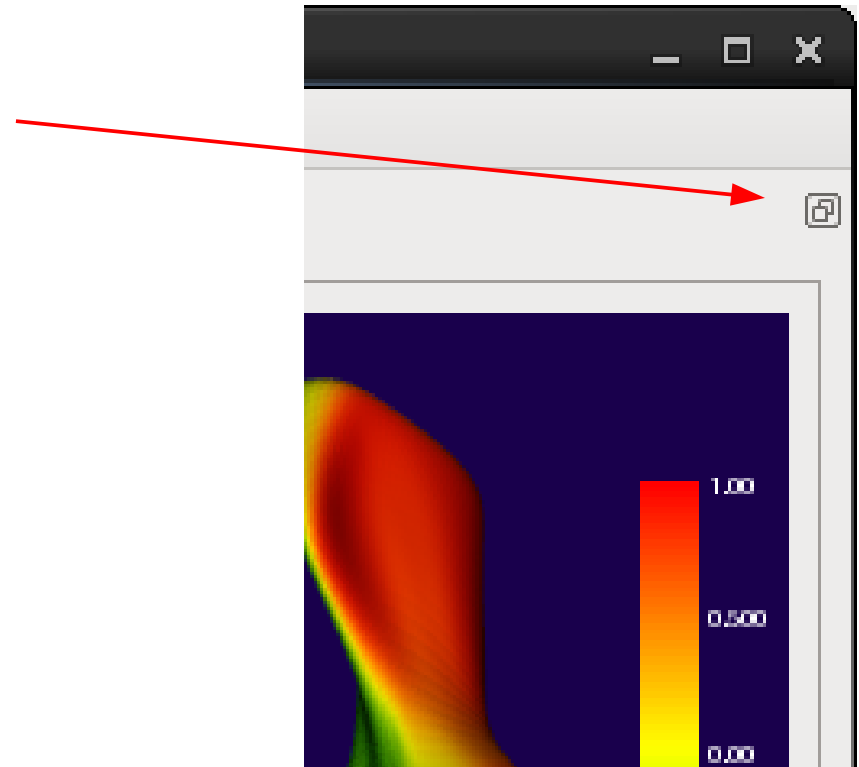
- **CTRL + Left Mouse** : Select or Unselect the window on which you clicked
- **CTRL + A** : Select All
- **ESC** : Unselect All



Interacting with surfaces

You can also display **only the Display Area** by detaching it from the main window, using the icon on the top right corner.

It allows you for example to display your meshes in **full screen**, after resizing the display area, if you only intend to interact with the surfaces without using the tools or information below the Display Area.





ShapePopulationViewer

GUI : Information Box



File Information

The first tab of the Information Box displays information about the file such as its **name and directory**, but also information about the loaded surface : **number of points and cells**.

If you select more than one surface at a time, you will not be able to get any information about them. You will have to select only one mesh to get its information.

The screenshot shows the 'File information' tab of the Information Box. The 'File information' tab is highlighted with a red circle. The fields are: Filename: 1.vtk, Directory: /ShapePopulationViewer/Data/DataTest, Number of Points: 4002, and Number of Cells: 8000.

The screenshot shows the 'File information' tab of the Information Box. The 'File information' tab is highlighted with a red circle. The filename field contains the text '2 surfaces selected, select only one' and is also circled in red. The other fields are empty: Directory, Number of Points, and Number of Cells.



Attributes/Ranges

The second tab of the Information Box displays the **scalar attributes names and ranges**.

- If different surfaces are loaded, **only the common attributes will be displayed**.
- When you select multiple meshes, the ranges displayed will be computed between the **minimum and the maximum among all the selected surfaces**

File information		Attributes/Ranges	
Name		Range	
Area		[0 ; 100]	
Area_Original		[0 ; 219.85]	
Area_Partial		[0 ; 5.9412]	
Normals		[-0.99994 ; 1]	
Radius		[0 ; 100]	
Radius_Original		[1.423e-13 ; 8.309]	

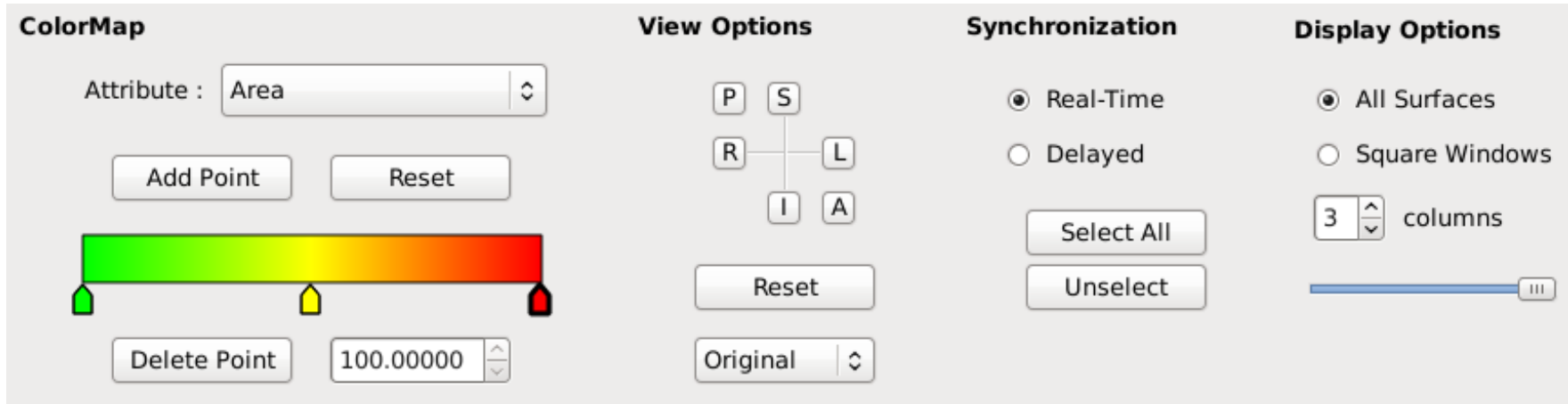


ShapePopulationViewer

GUI : Tool Box



Tool Box



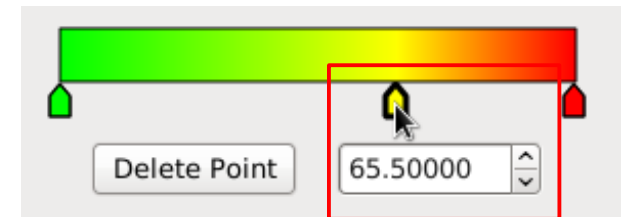
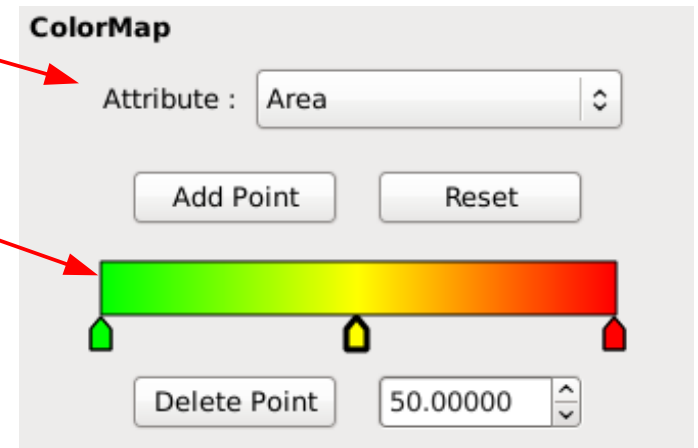
The Tool box includes the following functionalities :

- **Colormap** : Modify the gradient on the meshes
- **View Options** : Basic interactions with the camera
- **Synchronization** : Customize your interaction with the surfaces
- **Display Options** : Customize how surfaces are positioned in the Display Area



Colormap (1/3)

- **Attribute ComboBox** : you can define which scalar values you want to display.
- **ScalarBar** : displays the gradient computed between the computed range.
- **Add point** : add a new color arrow to the colorbar.
- **Reset** : reset the gradient to a pre-defined one.
- **Delete point** : pressing this button will delete the selected arrow and its color.
- **Value** : displays the scalar value associated to the selected arrow, between min and max.

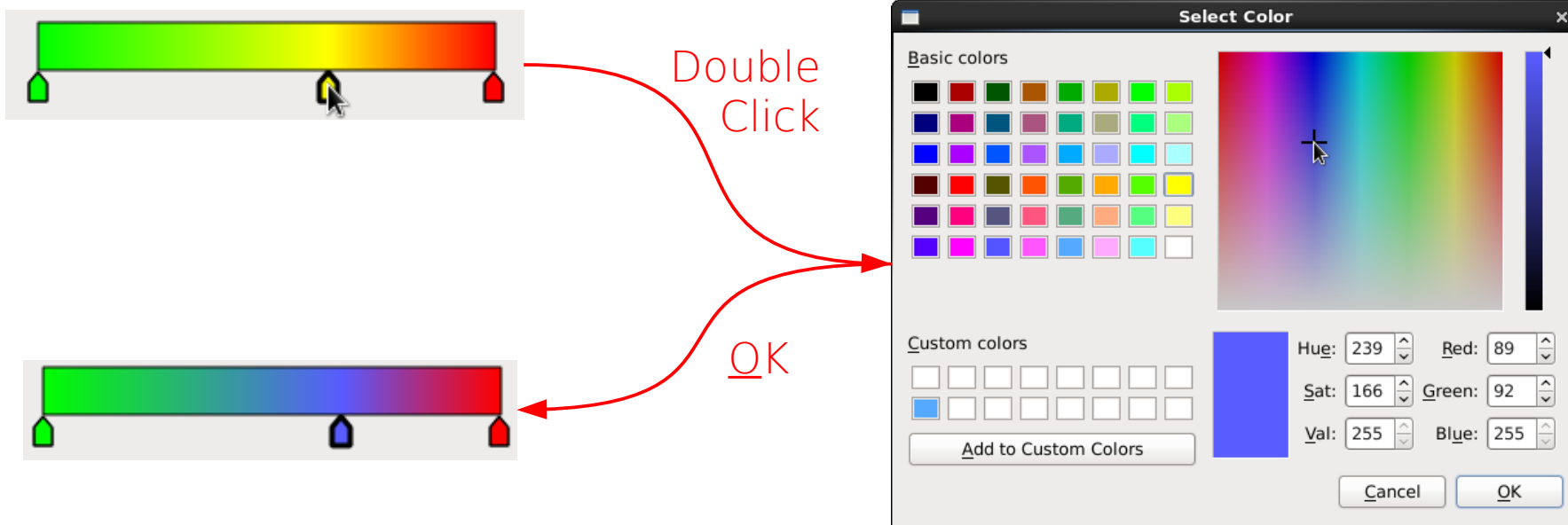


To modify its **scalar value** associated, **grab and move** it, or modify the value box directly.



Colormap (2/3)

- To modify a **color** associated with an arrow, simply **double-click** on it.
- A **colorPicker** window will appear, allowing you to select the desired color.





Colormap (3/3)

**A colormap is relative to an attribute and its range,
not to a mesh.**

*If you select multiple meshes, the display properties
of the first selected mesh will be used for all the other selected meshes
(camera position, displayed attribute, colormap)*



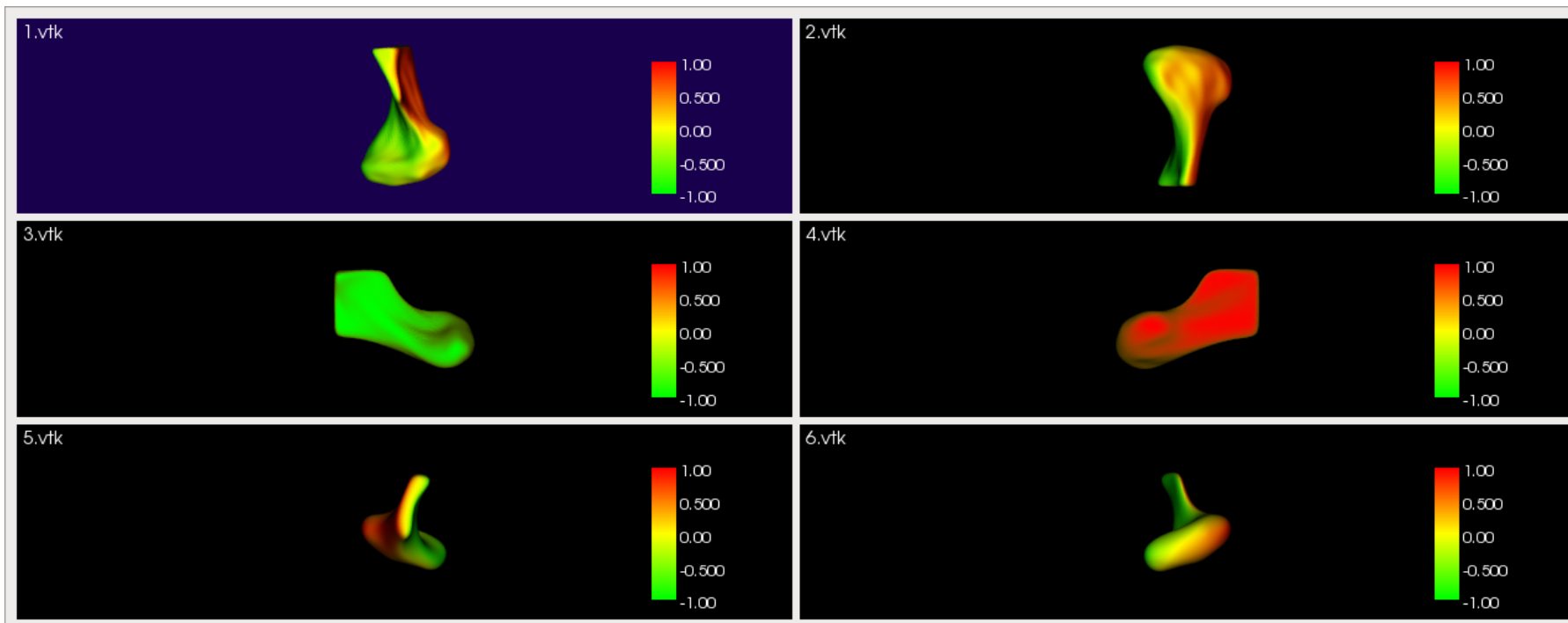
View Options (1/4)

- **Axis Buttons** : these 6 buttons allow the user to position the camera along the respective axis
- **Reset** : reset the focal point position and distance to the camera
- **Original/Centered ComboBox** : define the position of the surfaces





View Options (2/4)



Axis buttons : Example of the 6 camera positions :

Superior
Sagittal Right
Posterior

Inferior
Sagittal Left
Anterior



View Options (3/4)

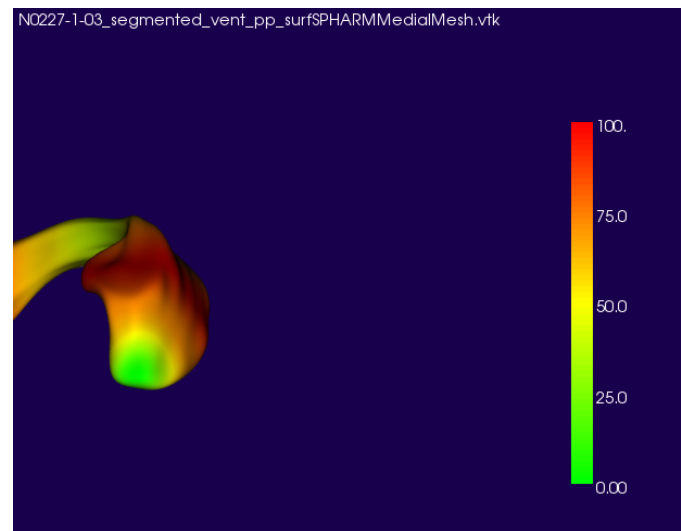
Reset Button :

Reset the focal point position (1)

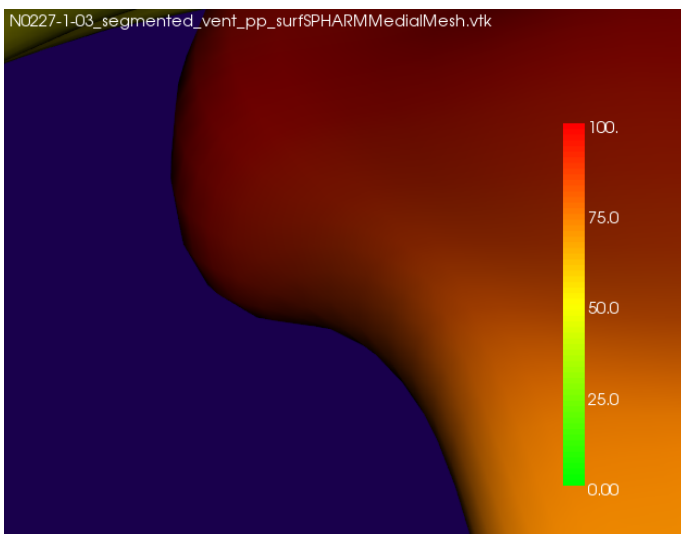
> modified by pressing the middle mouse button & dragging

Reset the camera distance (2)

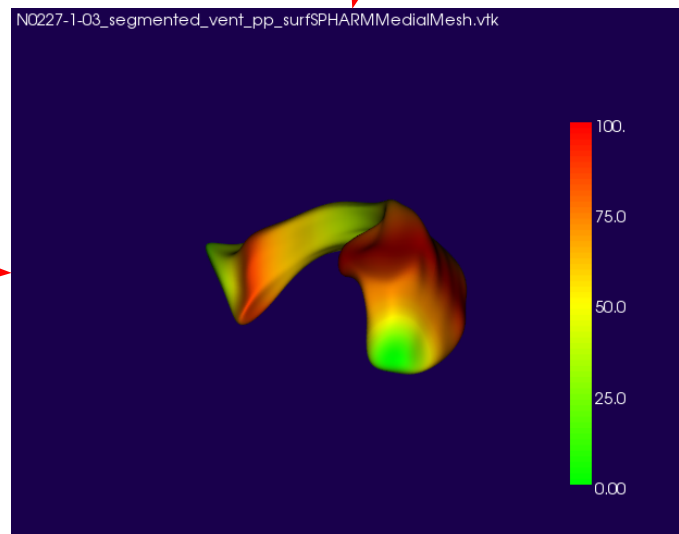
> modified by rolling the middle mouse button



(1)



(2)





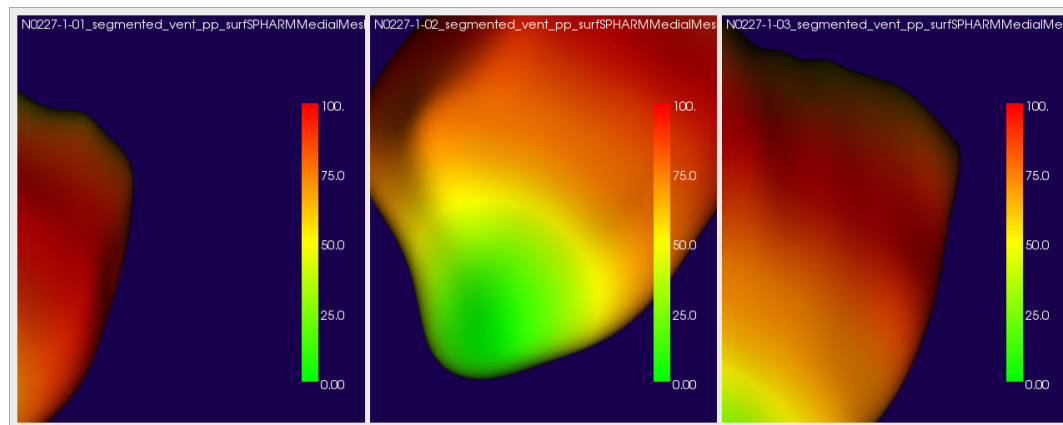
View Options (4/4)

Original/Centered ComboBox :

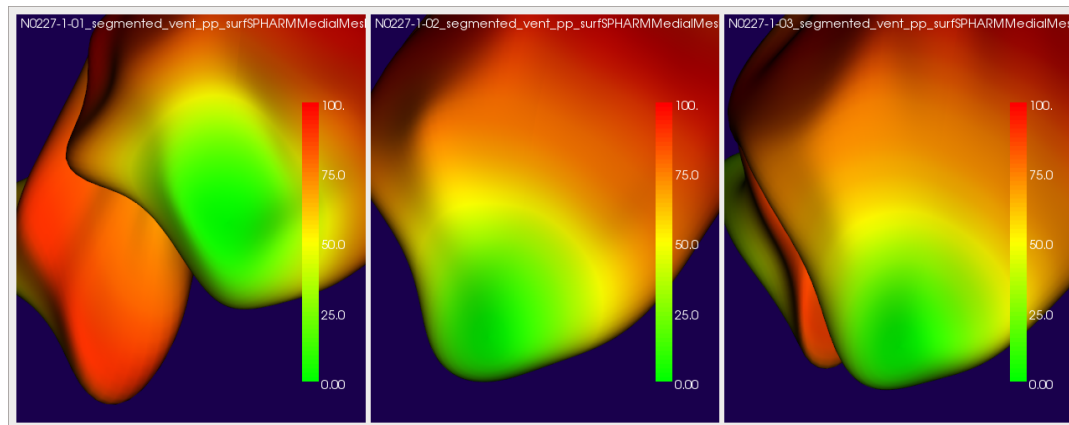
When loaded, surfaces are displayed with the position values associated to every vertex, as stored in the vtk file.

Sometimes it can be laborious to compare precise areas using this configuration, because surfaces are not aligned.

When a mesh is **“centered”**, using this tool, **its center of gravity is translated to the origin**, allowing the user to properly visualize and compare different surfaces.



Original  Center 





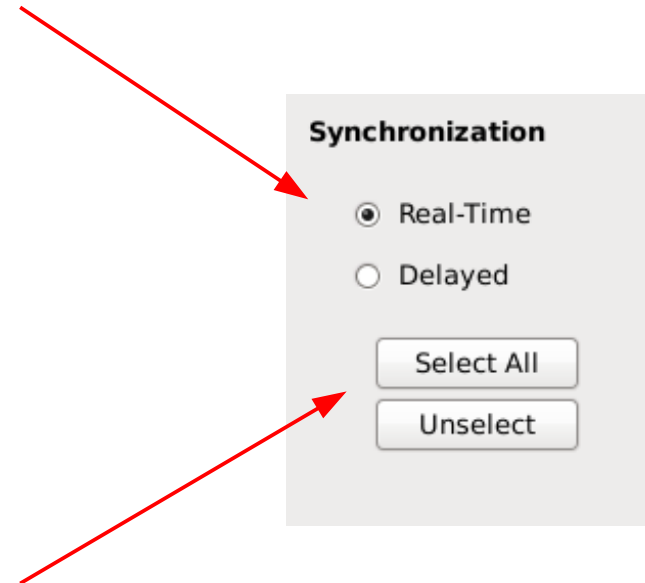
Synchronization

- **Real-Time/Delayed :**

- *Real-Time* : all the selected meshes are rendered at the same time as the one being interacted with.

- *Delayed* : all the selected meshes are rendered once the interaction with one of them is finished. This option is useful when processing a large number of meshes/points.

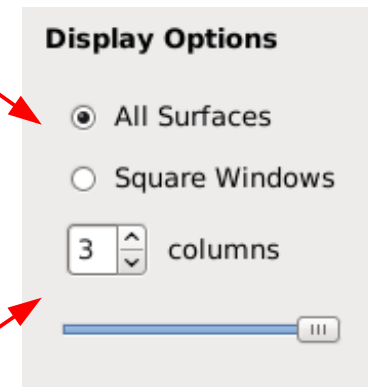
- **Select All** : select all the meshes (Ctrl+A)
- **Unselect** : unselect all the meshes (ESC)





Display Options

- **All Surfaces / Square Windows :**
 - *All Surfaces* : all the meshes will be displayed to fit in the display Area, modifying their render-window size.
 - *Square Windows* : the meshes render-windows will be square no matter what, their disposition will then depend on the chosen number of columns. You will be able to scroll in the Display Area to visualize all the meshes.
- **Columns** : Organizes the windows in the indicated number of columns.





ShapePopulationViewer

GUI : Advanced Options



Option Menu

The menu bar contains an “Option” Menu which includes three sub-options :

- **Camera configuration** : modify the camera configuration values, load and save them into a .pvcc file.
- **Load & Save Colorbar** : load & save the arrows colors and positions into a .spvcm file.
- **Window colors** : customize the background and text colors used in the render windows.

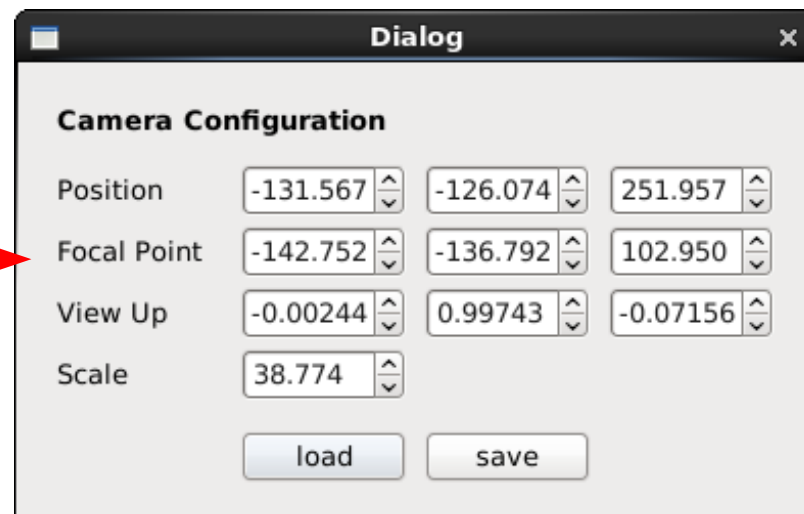


Camera Configuration

The camera Configuration tab launches a dialog window to configure your camera.

You can :

- **directly modify** the Position, Focal Point, View Up vectors and the Scale scalar of the camera.
- **Load or Save** a .pvcc file (ParaView camera configuration)



Be careful about your mesh position : Centered or Original.

This will influence the camera configuration values.



Load & Save Colorbars

```
<?xml version="1.0" encoding="UTF-8"?>
<SPVColorMap description="ShapePopulationViewer ColorMap" version="1.0">
  <colormap name="reset" points="3">
    <colorpoint index="0">
      <position>0</position>
      <R>0</R>
      <G>1</G>
      <B>0</B>
    </colorpoint>
    <colorpoint index="1">
      <position>0.5</position>
      <R>1</R>
      <G>1</G>
      <B>0</B>
    </colorpoint>
    <colorpoint index="2">
      <position>1</position>
      <R>1</R>
      <G>0</G>
      <B>0</B>
    </colorpoint>
  </colormap>
</SPVColorMap>
```

You can save and load colorpoints to use custom colorbars quickly.

The file format spvcm stands for “ShapePopulationViewer ColorMap”.

It is a very basic xml file which stores the points, their relative position between 0 and 1, and their color components, RGB, between 0 and 1.



Drag & Drop

Drag & Drop also works with the camera and colormap configurations files.

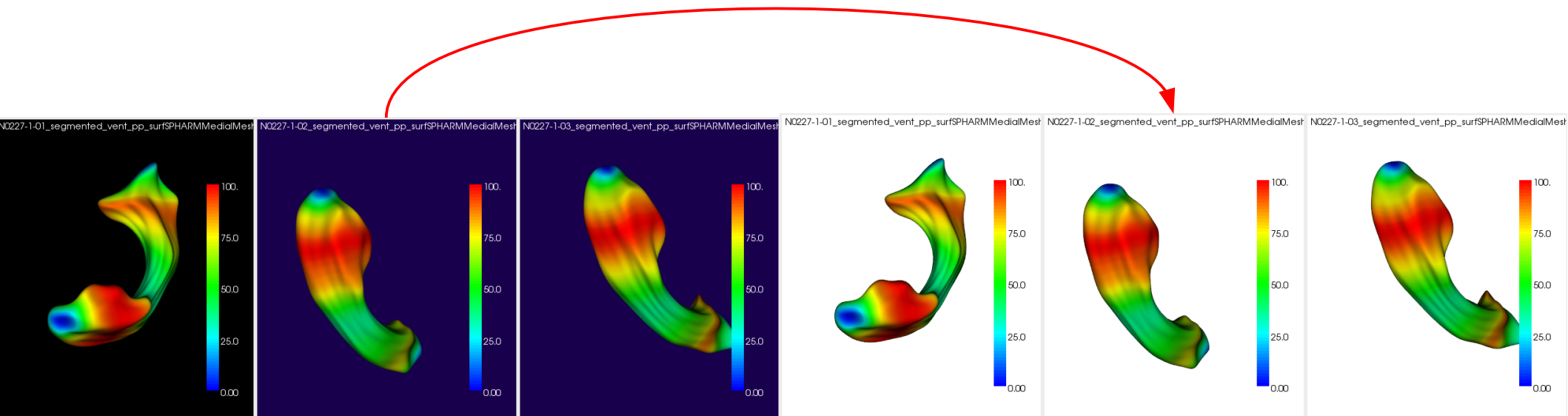
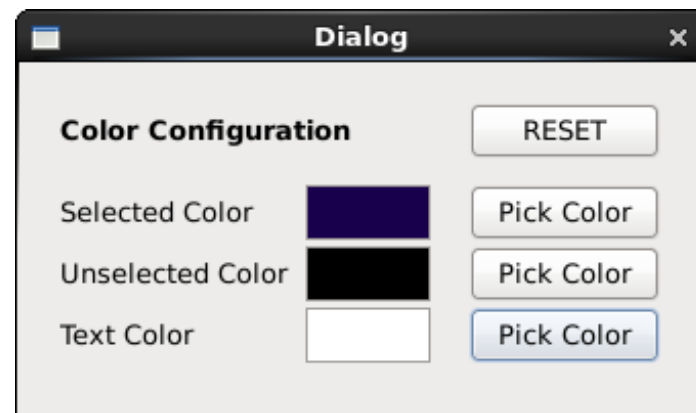
You can drop .pvcc files and .spvcm files in the MainWindow to load the camera configurations and the colorpoints easily.



Window Colors

The Window colors tab launches a dialog window to customize the **background and text colors**.

It is useful for example if you want to display white backgrounds, to present results in a publication :





ShapePopulationViewer

Command Line Interface



Command Line Interface

ShapePopulationViewer can be launched from the command line to directly load **vtk files/directory**, and **configuration files** (colormap and/or camera).

You can visualize all the options by running :

```
$ ./ShapePopulationViewer --help
```



Command Line Interface

`-v <std::vector<std::string>>, --vtkfiles <std::vector<std::string>>`

.vtk Input files (accepted multiple times)

`-d <std::string>, --directory <std::string>`

Input directory

`--csv <std::string>`

.csv Input file

`-c <std::string>, --camera <std::string>`

.pvcc Camera Configuration File

`-g <std::string>, --gradient <std::string>`

.spvcm Colormap Configuration File



ShapePopulationViewer

Help & Contact



Help

Wiki

www.nitrc.org/plugins/mwiki/index.php/shapepopviewer:MainPage

Forums

www.nitrc.org/forum/?group_id=759

Bug & Features Tracker

www.nitrc.org/tracker/?group_id=759



Contact

For additional remarks or questions, please email :

girault@mail.unc.edu