# *BRainbow*

**BRainbow**

NEURONIC
Copyright © Neuronic S.A. 2002-2004

**User Manual**

**Version 1.0**

NEURONIC S.A.

**2007**

## DISCLAIMER

The **BRainbow** software is designed for use in medical research, it has not the status of a medical product, i.e. it is not allowed to use it directly or indirectly for medical prevention, diagnostic and/or treatment of humans. It is solely for academic use in the field of scientific research. NEURONIC S.A. does not grant warranty or assume liability for the results of using the software.

The content of this document is the intellectual property of NEURONIC S.A., and is subject to change without specific notification. NEURONIC S.A. does not grant warranty or assume liability for the correctness of individual statements herein. Nor does NEURONIC S.A. enter into any obligation with regard to this document.

Any trademarks mentioned in this document are the protected property of their rightful owners.

# Index

# Introduction

**BRainbow** is Windows MDI (Multiple Document Interface) software application for visualizing functional and anatomical surfaces extracted from brain images. **BRainbow** process surfaces in SRX format (see iMagic User Manual). The following figure shows the different elements of the application on screen.
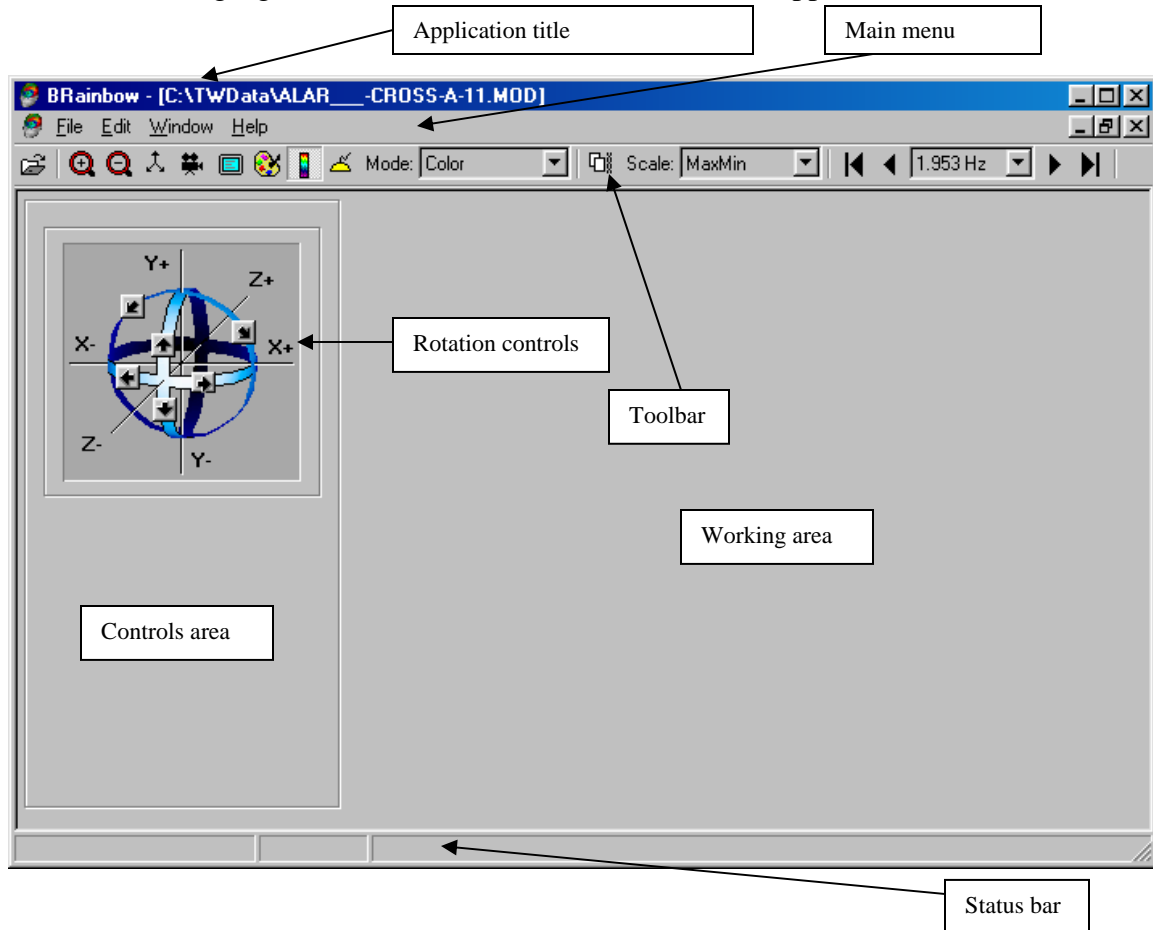


In the application title, the active data filenames will be shown. In the toolbar are located buttons for the most frequently used commands. The status bar, depending on the loaded data, shows diverse information about the data being processed: data value, measure name, etc.

In the working area the data loaded is shown. The rotation controls allow by pressing the buttons the surface rotation, and are always present whenever surface data is opened. Depending on the loaded data, the controls area shows different elements for handling the data.

## Elements of the toolbar

| | |
|---|---|
| ☞ | Open map |
| ⊕ | Zoom in |
| ⊖ | Zoom out |
| ⋏ | Reset to default position and size |
| 🎥 | Create movie |
| ▣ | Full screen |
| 🎨 | Change palette |
| ▮ | Show palette bar |
| ◿ | Show electrodes |
| Mode: Color ▾ | Change mode type |
| 🗗 | Yoke windows |
| Scale: MaxMin ▾ | Change scale type |
| ⏮ | First temporal data |
| ◀ | Previous temporal data |
| 1.953 Hz ▾ | Change temporal data |
| ▶ | Next temporal data |
| ⏭ | Last temporal data |

A detailed explanation of these commands will be given in the following chapters.

## Mouse commands

The left button when pressed and dragged allows the rotation of the surface interactively. The right button when pressed and dragged allows the panning of the surface interactively. When rolling the wheel or pressing simultaneously both buttons while moving the mouse, the surface is zoomed in/out. Moving the mouse over a map or tomographic window will show in the status bar the current value under the mouse cursor.

# Description of commands

## Open map 📂

This command loads the map to plot. When activated, the following dialog is shown.
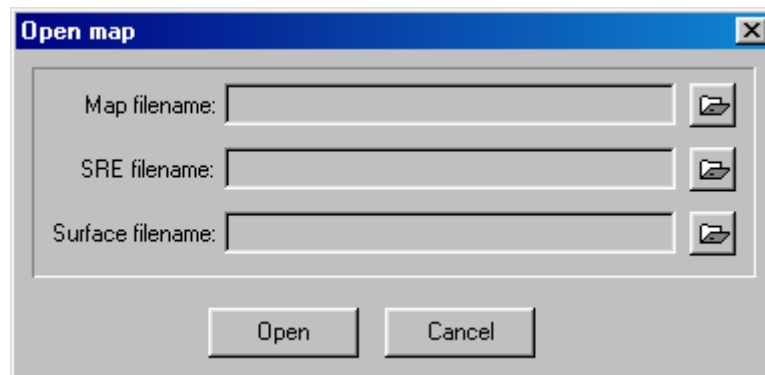


The Map filename edit box allows setting the map filename. **BRainbow** loads MOD files (NEURONIC S.A. format) , ASCII files and binary files.

The ASCII files in order to be plotted must comply with the following rules. The first line of the file will have the electrode labels separated by spaces. In the following lines will appear the values corresponding to the maps to plot one map per line. In the line every value must be separated by spaces. The number of values per line must match the number of labels in the first line. Example:

```
Fp1 Fp2 F3 F4 C3 C4 P3 P4 O1 O2 F7 F8 T3 T4 T5 T6 FZ CZ PZ¶
5.4828  5.5790  5.6875  5.9411  5.6107  5.9344  5.9148  5.8339  5.9785
6.1317  5.6740  5.7567  5.7541  5.8708  5.9028  6.1042  5.7156  5.7057
5.9162¶
5.9344  5.9148  6.1042  5.7156  5.8339  5.4828  5.5790  5.6875  5.9411
5.6107  5.9785  6.1317  5.6740  5.7567  5.7057  5.9162  5.7541  5.8708
5.9028¶
```

In the example there are two maps to plot, two lines plus the first one with the labels identifying the electrodes.

The binary file to load must be a sequence of floating point numbers in simple precision (4 bytes). In this case no information about which electrodes are present in the file is given, so **BRainbow** will assume the file will have the same number and order of channels given by the SRE file (see below). The file can have as many maps as needed.

The Surface filename edit box allows setting the SRX filename. The SRX (surface) file (see iMagic User Manual) stores the surface used for plotting the map.

The SRE filename edit box allows setting the SRE filename. The SRE file (see iMagic User Manual) contains the information needed for establishing the correspondence between the surface (SRX file) and the electrode montage. The value information to be mapped is the intersection of the electrode montage defined in the SRE file and the electrode montage (electrode labels) appearing in the Map filename. It means that only the electrode that appears in both files is mapped.

When installed **BRainbow** copies one surface (NORS.SRX) which corresponds to the Montreal Neurological Institute average brain and 3 SRE files (NORS1020, NORS120, NORSBC) corresponding to the standard 10/20 system, the 120 channel NEURONIC system and BrainCap 74 channel respectively. The files are placed in the directory where **BRainbow** was installed. The SRE and SRX files can be obtained with the **iMagic** system.

Once the filenames are set, the Open button loads the map in the working area. If the filename has more than one measure for mapping, an auxiliary dialog is presented for selecting the desired measure, in this case the measure's name will be indicated in the status bar.
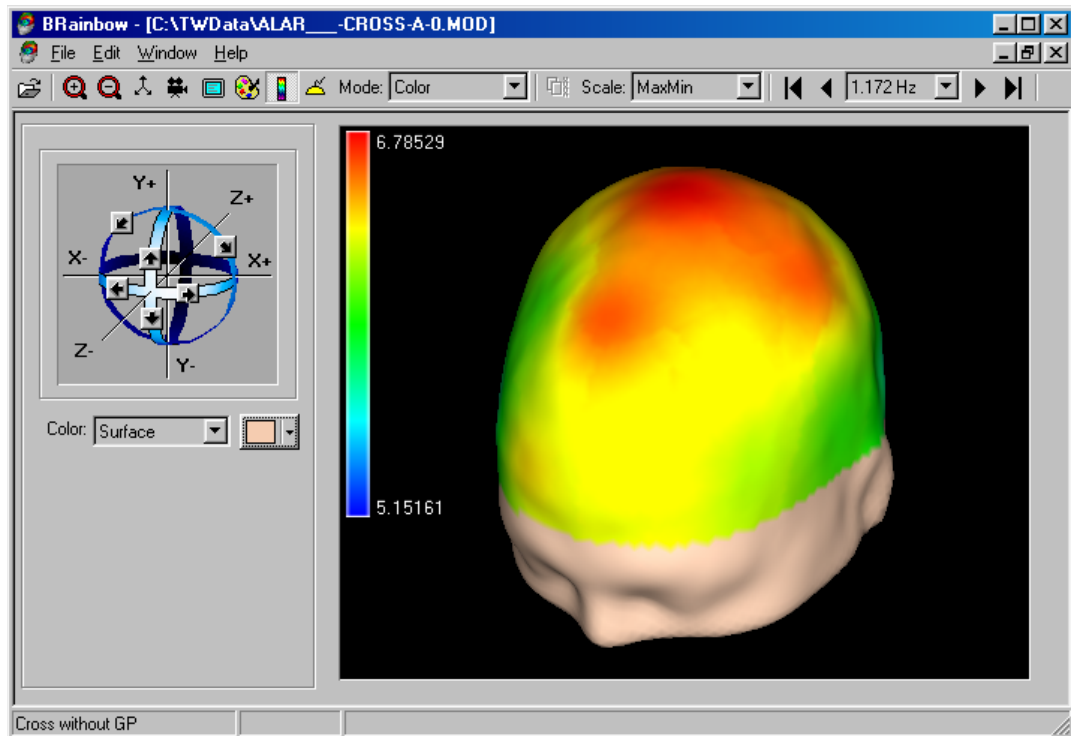


When maps are loaded the Controls area show the Color selection element:
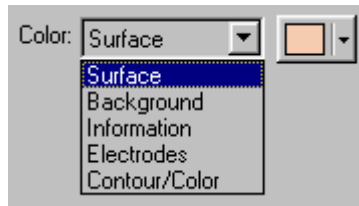
With this combo box a color selection for different elements of the working area is possible. The color settings established here are valid only for the active window.

## Open surface field

This command loads a surface field (tomography) for plotting. When activated, the following dialog is shown.
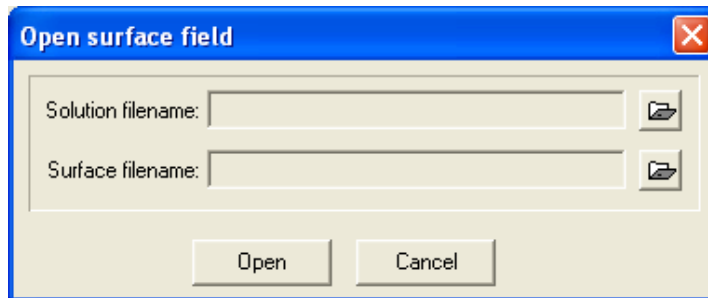


The Solution filename edit box allows setting the solution filename. **BRainbow** loads ASCII files (any extension) in two different modalities: fields over vertex and fields over triangles. These files must have a value per line corresponding to the field (tomographic solution) calculated for every vertex or triangle of the surface. The file will have as many values (lines) as vertex (triangles) in the surface multiplied by the number of solutions in the file. In order to set the modality (over vertex or over triangles) the solution filename must be given by using the browse button and the corresponding standard open dialog. In the dialog the **Files of type** combo box must be set accordingly.

The Surface filename edit box allows setting the SRX filename. The SRX (surface) file (see **iMagic** User Manual) stores the surface used for plotting the field (tomographic solution).

Once the filenames are set, the Open button loads the field (tomographic solution) in the working area.
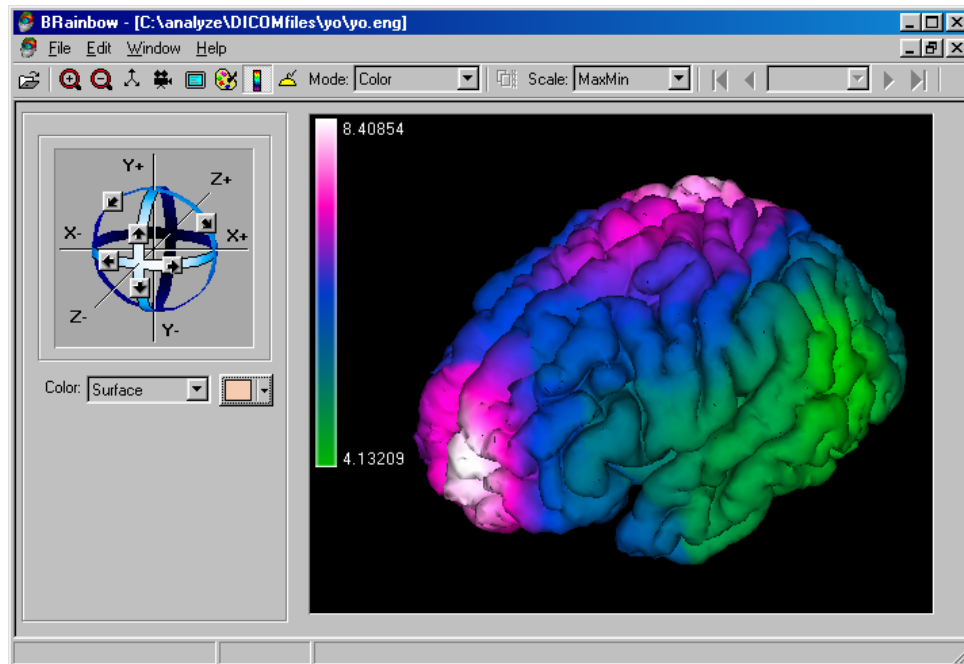
When fields (tomographic solutions) are loaded the Controls area shows the Color selection element (see **Open Map** command)

## Open fiber tracking

This command can load an image volume and/or surfaces contained on it and also fiber bundles calculated (fiber tracking) for this image volume. When activated, the following dialog is shown.
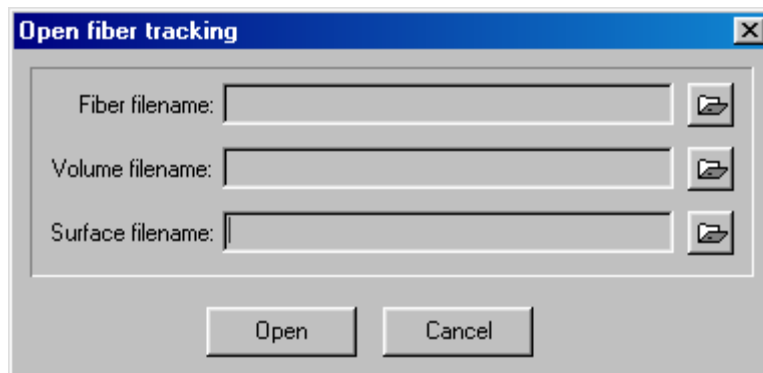


The Fiber filename edit box allows setting the fiber bundle filename(s). **BRainbow** loads TXT files (ASCII files). These files have 3 values per line corresponding to X,Y,Z coordinates of the track point. The file will have as many lines as track points in the fiber bundle. It is possible to load multiple files at once. With the command **Add fiber bundle** is possible to add more fibers later.

The Volume filename edit box allows setting the volume filename. The VOX (volume) file (see **iMagic** User Manual) stores the volume image. The volume

image will be presented by means of a combination of the three orthogonal planes: axial (Plane Y), sagittal (Plane X) and coronal (Plane Z).

The Surface filename edit box allows setting the SRX filename. The SRX (surface) file (see **iMagic** User Manual) stores the surface extracted from the volume. With the command **Add surface** is possible to add more surfaces later.

It is user responsibility the coordinates matching between the volume, surface(s) and fiber bundle(s). At least a volume or surface filename must be set to process this command.

Once the filenames are set, the Open button loads the files in the working area.
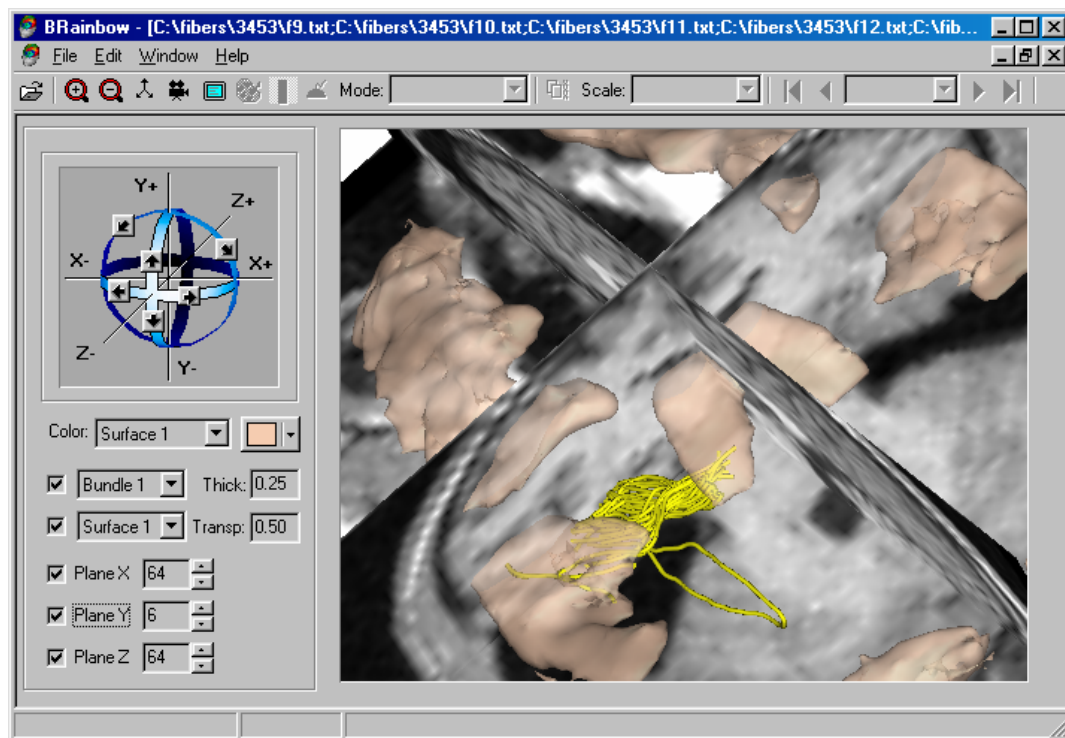


When files are loaded the Controls area can show other elements depending on the data loaded.

If fiber bundles are loaded the following elements are presented:



The combo box allows selecting the active fiber bundle. The check box to the left indicates and set visibility of the active bundle. The **Thick** edit box sets the thickness of the active bundle.

If surfaces are loaded the following elements are presented:

The combo box allows selecting the active surface. The check box to the left indicates and set visibility of the active surface. The **Transp** edit box sets the transparency of the active bundle. This is a coefficient between 0.0 and 1.0, 0.0 means full transparency (not visible) and 1.0 means complete opaque.

If a volume is loaded the following elements are presented:



The check box to the left indicates and set visibility of the named orthogonal plane: sagittal (Plane X), axial (Plane Y) and coronal (Plane Z). The edit box to the right sets the exact plane coordinate to show. In the Color selection element, the fiber bundles and surfaces will be present for selecting their color.

## Open NIRs

This command can load NIRs (Near InfraRed spectroscopy) data for plotting. When activated, the following dialog is shown.
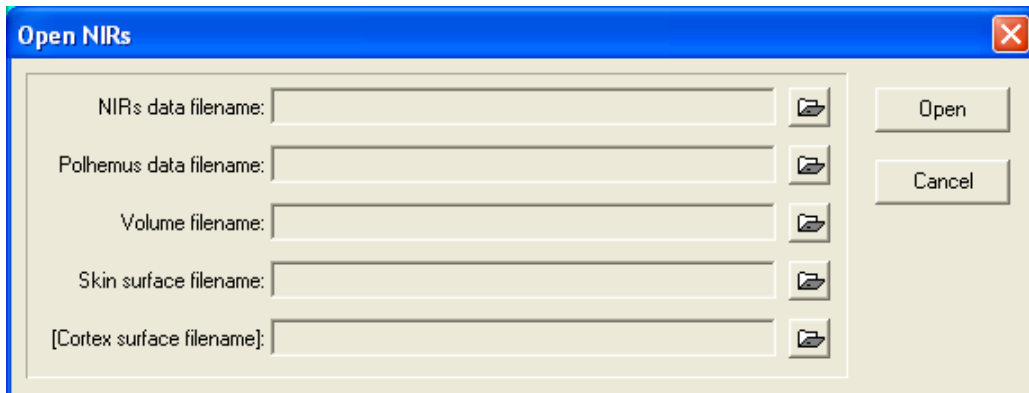


The NIRs data filename and Polhemus data filename edit boxes allow setting the filenames. **BRainbow** loads Hitachi NIRs and Polhemus data files (ASCII files). It's user responsibility to match NIRs data with Polhemus data according to the probes configuration used.

The Volume filename edit box allows setting the volume filename. The VOX (volume) file (see **iMagic** User Manual) stores the volume image. The volume image will be used for fitting Polhemus measurements to the image itself.

The Skin surface filename edit box allows setting the SRX filename. The SRX (surface) file (see **iMagic** User Manual) stores the skin surface extracted from the

volume. Over this surface the NIRs data will be plotted. The optional Cortex surface filename allows to load a second surface for plotting over the cortex instead of the skin.

Once the filenames are set, the Open button loads the files in the working area.
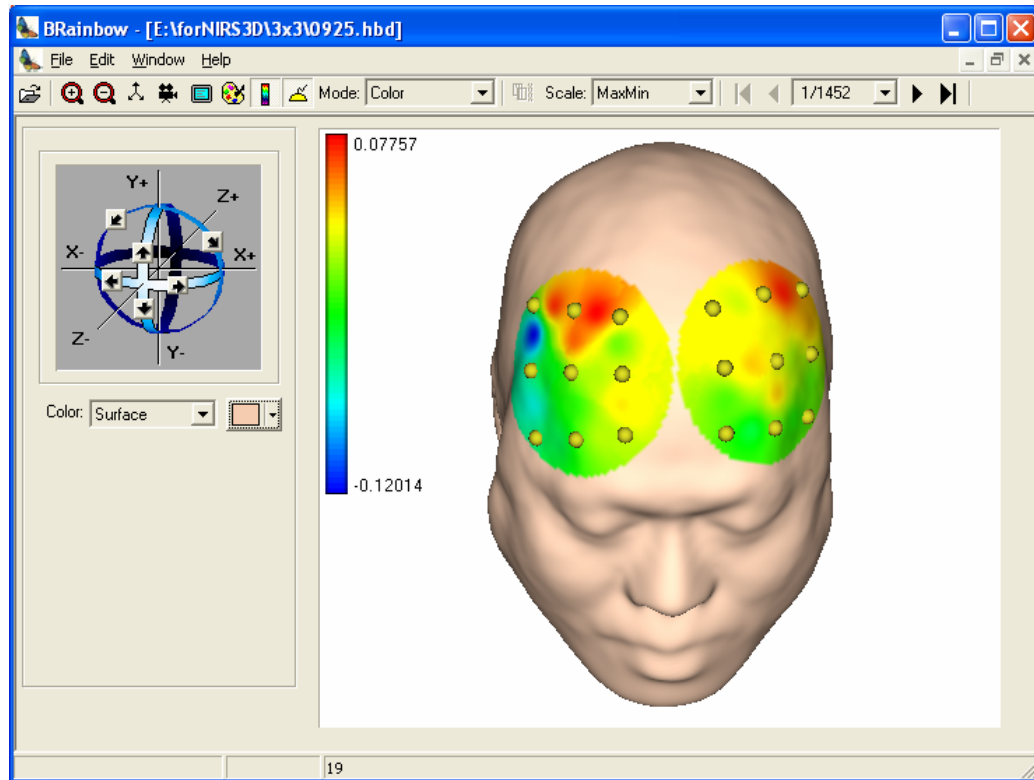


After loading the data the Controls area shows the Color selection element (see **Open Map** command)

## Open connections

This command can load connections data for plotting. When activated, the following dialog is shown.
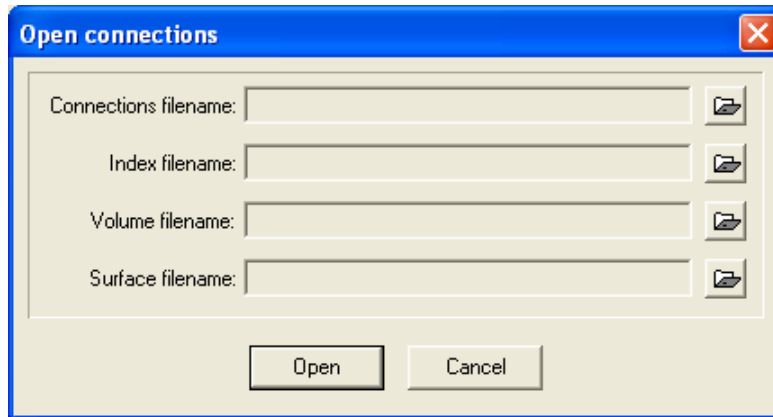
The Connections filename edit box allows setting the connections data filename. **BRainbow** loads TXT files (ASCII files) containing the connection information. These files have 3 values per line corresponding to origin index, destination index and connection strength respectively. The origin and destination indexes are integer numbers indicating the points in the index file. The file will have as many lines as connection lines.
Example:

```
13 14 5.4828
2 26 5.5790
11 1 5.6875
```

In the example there are 3 connections to plot. The first one going from the point 13 to the point 14. It means that the connection will be plotted from the 14$^{th}$ coordinate point in the index file to the 15$^{th}$.

In the Index filename edit box the file containing the index is set. The index file is an ASCII file with 3 values per line indicating the X,Y,Z coordinate point respectively. The file will have as many lines as coordinate points.

The Volume filename edit box allows setting the volume filename. The VOX (volume) file (see **iMagic** User Manual) stores the volume image. The volume image will be presented by means of a combination of the three orthogonal planes: axial (Plane Y), sagittal (Plane X) and coronal (Plane Z).

The Surface filename edit box allows setting the SRX filename. The SRX (surface) file (see **iMagic** User Manual) stores the surface extracted from the volume. With the command **Add surface** is possible to add more surfaces later.
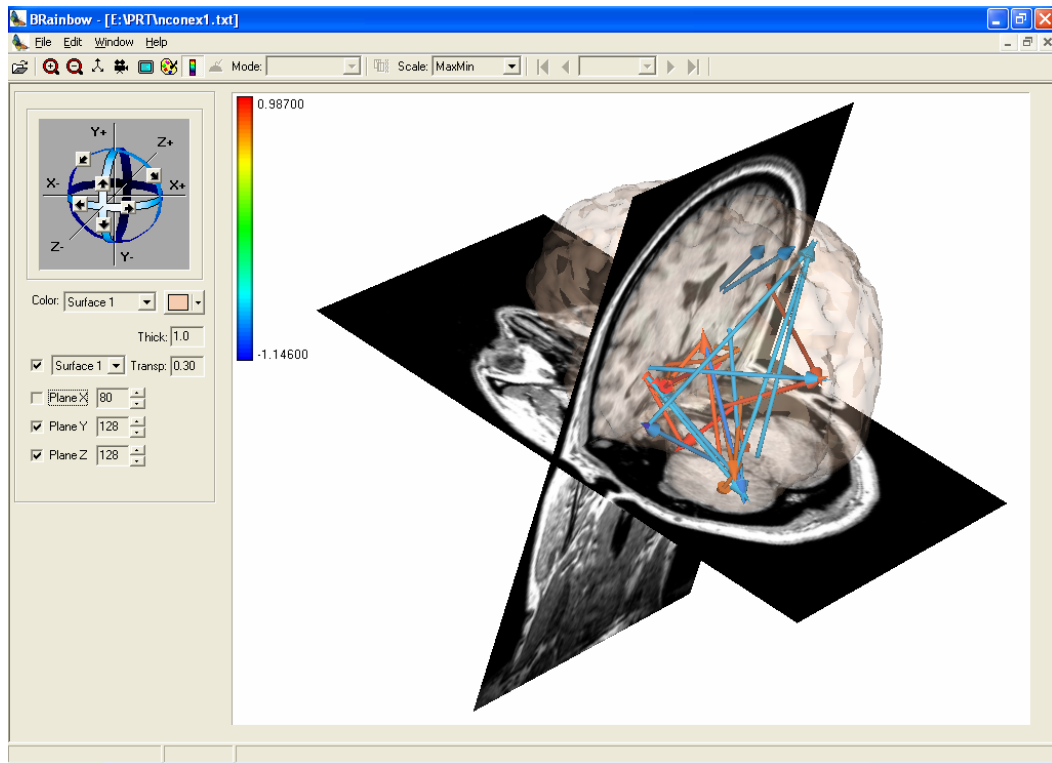
Once the filenames are set, the Open button loads the files in the working area. At least a volume or surface filename must be set to process this command. The connections are represented by arrows coming from the origin point to the destination point.

When files are loaded the Controls area can show other elements depending on the data loaded.

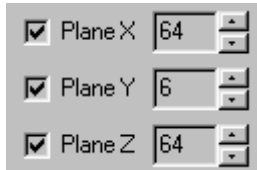If surfaces are loaded the following elements are presented:



The combo box allows selecting the active surface. The check box to the left indicates and set visibility of the active surface. The **Transp** edit box sets the transparency of the active bundle. This is a coefficient between 0.0 and 1.0, 0.0 means full transparency (not visible) and 1.0 means complete opaque.

If a volume is loaded the following elements are presented:

The check box to the left indicates and set visibility of the named orthogonal plane: sagittal (Plane X), axial (Plane Y) and coronal (Plane Z). The edit box to the right sets the exact plane coordinate to show. In the Color selection element, the surfaces will be present for selecting their color.

The edit box Thick sets the thickness of the connections arrows.

Thick: 0.25
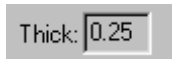
## Add fiber bundle

This command allows adding fiber bundles to the already loaded with the **Open fiber tracking** command. When activated, a standard Windows dialog is presented for selecting the fiber bundle file(s) to add.

## Add surface

This command allows adding surfaces to the already loaded with the **Open fiber tracking** and **Open connections** command. When activated, a standard Windows dialog is presented for selecting the surface to add.

## Add geometric object

This command allows adding geometric objects to the data already plotted by the commands **Open fiber tracking** and **Open connections**. When activated, the following dialog is shown.
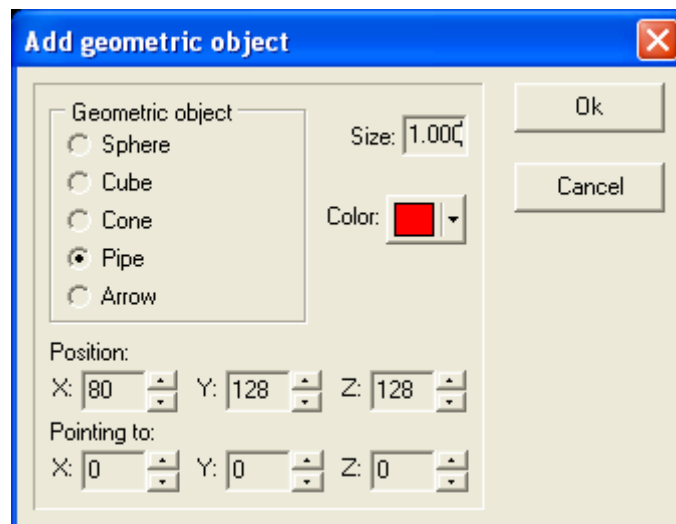


In the group **Geometric object** the desired object is selected. The edit box **Size** sets the object's size. Depending of the object type, the size is interpreted as radius, height, etc. The **Color** selector allows to set the object's color. The

**Position** controls set the object center coordinates. When selecting pipes or arrows the **Pointing to** controls set the destination point coordinates, while **Position** set the origin.

After adding the object, in the Controls area will appear a new control for selecting and editing the geometric objects present in the display.
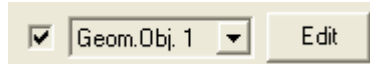


## Properties

Shows a dialog with the characteristics of the active window depending of the data loaded. If the active window has a map loaded a window like the following is presented:
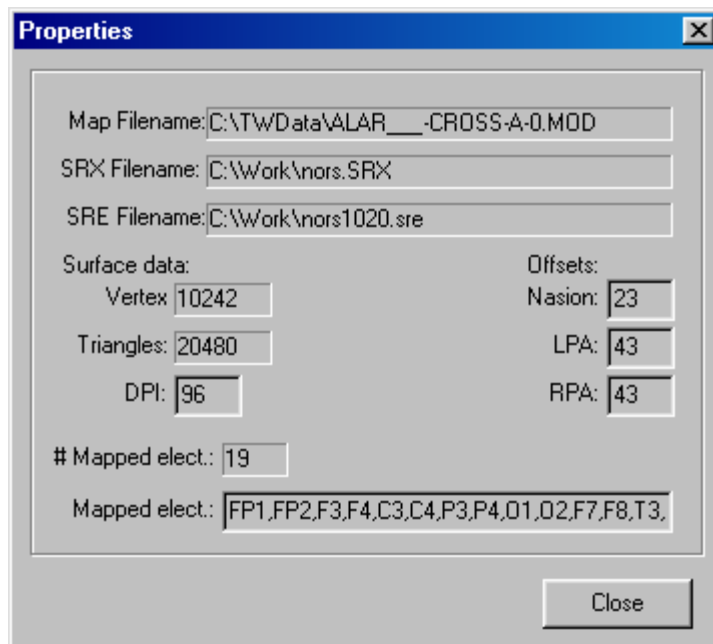


The fields related to delimitation of the map zone over the surface (Offsets: Nasion, LPA, RPA) can be edited in order to adjust the zone according with the montage used. The map zone is determined by a plane (given by those 3 points) intersecting the surface. When a value is set on these fields, that value is added to the Y coordinate of the given point.

The DPI field of the Surface group can be edited too for setting the dots per inch to achieve with the Copy commands (see **Copy** and **Copy to** commands). The other fields are shown only for informative purpose and are self-descriptive. When the active window is loaded with tomographic solutions or fiber tracking data, all the fields shown are only for informative purpose with the exception of the DPI field already explained.

## Close

Close the active window.

## Exit

Close the **BRainbow** application.

## Copy

Copies the contents of the active window to the Windows clipboard.

## Copy to

This command allows copying the content of the active window to a Windows bitmap file. When activated a dialog for selecting the name and format type (BMP or JPG) of the file appears.

## Set contour steps

This command set the number of contours to paint when the Contour or Contour/Color modes are used (see **Change Mode** type command)

## Set rotation step

This command set the step used when rotating around the axes using the rotation controls of the Controls area.

## Set labels

This allows to set labels for the current temporal sequence. When activated a dialog is presented for setting the parameters needed for generating automatically the sequence labels. There are 3 edit boxes: Initial Value, Sampling rate and Units. The Initial Value field allows to set by using digits the starting value for the sequence labels. Sampling rate will indicate the numeric value needed for obtaining the label for the next labels. The label value in a given instant is calculated by adding the label value in the previous instant with Sampling rate. The Units field will indicate the units used if any. The command is disabled if the active data loaded is not a temporal sequence command.

## OpenGL info.

This command shows a dialog with information about the OpenGL driver in use.

## Zoom in

This command zooms in the active window

## Zoom out

This command zooms out the active window

## Reset to default position and size

This command resets the data loaded in the active window to the default position (data "looking" to the user) and size.

## Create movie

With this command a movie can be generated. The resulting animation is stored in standard Windows format **.AVI**. When activated the following dialog is presented:
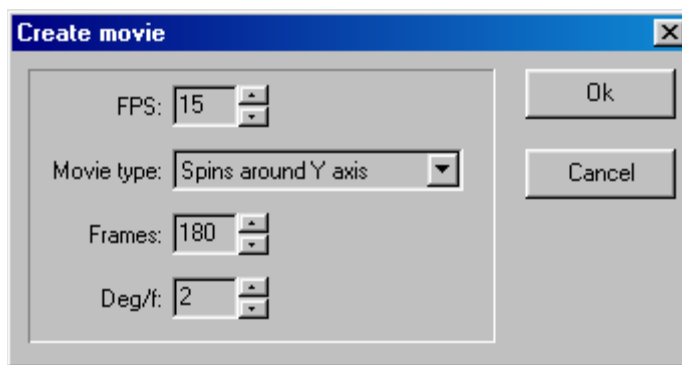
The FPS edit box sets the Frames per second ratio to be used in the movie. The Movie type combo box allows the selection of different movie types:

|  |  |
|---|---|
| **Spins around … axis** | This movie type generates rotations of the data loaded around one of the 3 possible axes. When selected, the fields Frames and Deg/f (degrees per frame) are presented. The Frames edit box sets the number of frames to generate in the movie. The Deg/f. edit box sets the degrees to rotate in every frame. |
| **Zoom in/out** | This movie type generates zooms (in or out depending on the option selected) of the data loaded. When selected, the field Frames is presented. The Frames edit box sets the number of frames to generate in the movie. The zooming carried out in every frame is fixed by the system and is similar to the zooming effect carried out by the **Zoom in/out** commands. |
| **Temporal** | This movie type generates a temporal animation of the data loaded. When selected, the fields From and To are presented. The From edit box sets the starting temporal instant from which the movie will be generated. The To edit box sets the final temporal instant of the animation. |

**Move … plane**  This movie type generates an animation of the selected plane movement. When selected, the fields Frames and Vox./f (degrees per frame) are presented. The Frames edit box sets the number of frames to generate in the movie. The Vox/f. edit box sets the number of voxels to move in every frame.

**User defined**  This movie type generates animations defined by the user. This animation type is made of a combination of the others movie types. The definition of this movie type is stored in MUX files, which can be created/edited through the Edit command shown in the dialog when this movie type is selected.  When the Edit button is pressed, the user is prompted to open an already existent MUX file for editing. If a non-existent filename is given a new MUX file will be created. Immediately after the following dialog is presented:
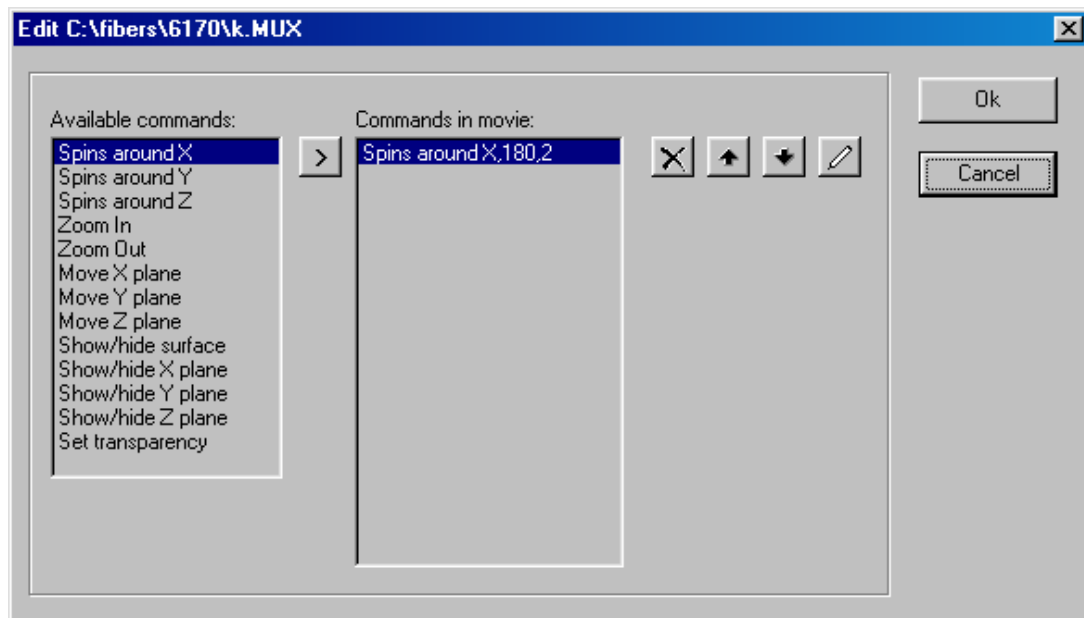


The left side shows a list box with the available commands. Using the  button, commands can be added to the Commands in movie list. The commands are added to the end of the list. Every time a command is added, a dialog is presented for setting parameters related to the command. The parameters for Spins, Zoom and Move commands were already explained before. The Show/Hide surface command presents a dialog for selecting the surface to show/hide. The Set transparency command presents a dialog with two fields: the Surface edit box for selecting the surface, and the

Transp. Edit box for setting the surface's transparency (see **Open fiber tracking** command, Controls area). The Show/hide plane commands don't need parameters.

The ⊠ button deletes the active command (command where the cursor is placed) in the Commands in movie list. The ▲, ▼ buttons move up and down in the list the active command respectively. The ✎ button shows the dialog for editing the active command parameters. When the Ok button is pressed the user animation definition is saved in the MUX file.

Once the movie type and its parameters are already defined, the Ok button can be pressed for editing the movie filename. The Cancel button aborts the movie creation. After selecting the movie filename the following dialog is presented:
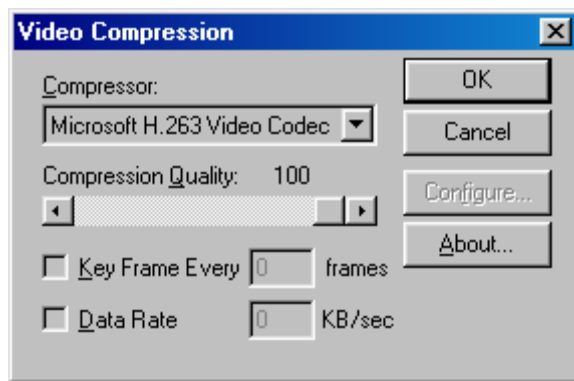


In this dialog a compressor (codec) for generating the movie is selected. The Compressor combo box lists all the codecs installed in the system. Depending on the codec selected additional parameters can be set. Depending on the codec characteristics and requirements, the movie is generated with more or less quality. **BRainbow** is not responsible for the performance of the codecs listed. Sometimes is not possible (by unknown reasons) to create the movie when using some codecs. The Full Frames (Uncompressed) codec always works and give optimal results from the quality point of view but the movies generated got a lot of space on disk.
Once the compressor is selected, the movie generation begins frame by frame on screen. During this process it is possible to abort the movie generation by pressing the ESC key.

## Full screen  ▣

When this command is activated, the working area is expanded to fill the whole screen. Use the **Esc** key to return to the normal mode.

## Change palette  🎨

Using this command is possible to change the active palette by selecting a new one or by editing the active one. When this command is activated a window is shown with several controls: a combo box list indicating the active palette's name, the active palette's preview, an option for interpolating the palette and the Edit button for editing the palette.

The palettes in **BRainbow** can use up to 256 colors.  If the palette does not have 256 colors (as usual) and the option for interpolating is set, the application linearly interpolates the present colors up to 256 to obtain the palette that is actually used for plotting the data. If the option for interpolating is not set, the palette will be used without interpolation.

By means of the combo box list it is possible to select any one of the available palettes. Once the window is closed with the Ok button the newly selected palette will become the active palette of the active window.  If the Edit button is pressed the window is enlarged adding new controls to change the palette.
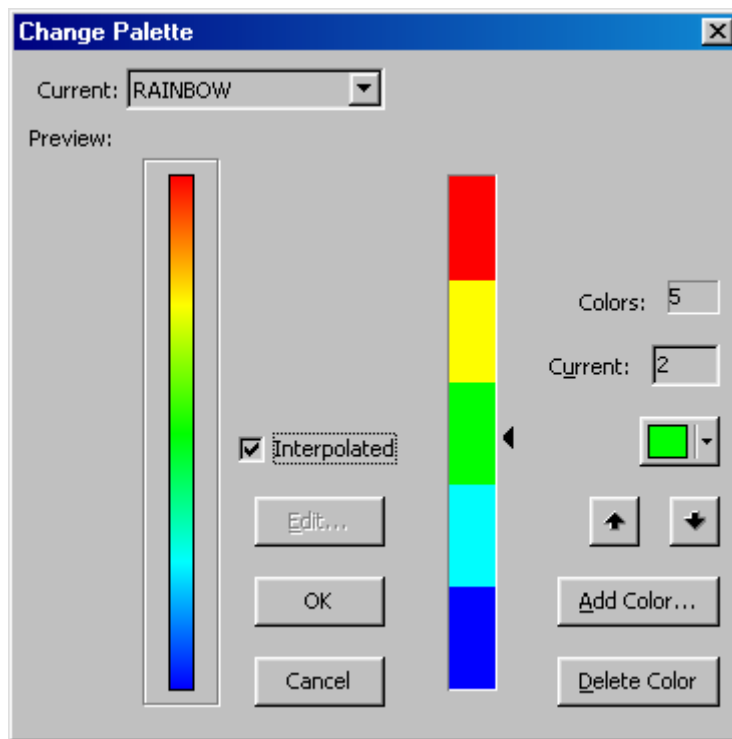


The first element that appears is a vertical bar divided by the quantity of real colors comprising the palette. The colors are denoted in bottom up order from 0 to the number of colors-1. The number of colors is indicated in the Colors box. Dragging the mouse over the cursor ( ◀ ) to the right of this bar sets the active color. The active color can also be selected with the Current edit box.

There are also 5 buttons for editing the palette:

This button changes the active color. When pressed, opens the standard Windows dialog for selecting colors.

This button moves the active color up in the palette. It is disabled when the active color is the last one in the palette.

This button allows moving the active color down in the palette. It is disabled when the active color is the first one in the palette.

This button adds a new color in the palette. When pressed, opens the standard Windows dialog for selecting colors. Once chosen, the new color becomes the active color and is inserted after the previously active color. It is disabled when the palette reaches the maximum of 256 allowed colors.
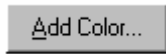
This button erases the palette's active color. The following color becomes the new active color. This button is disabled when the palette reaches the minimum of 2 colors.

When pressing the Ok button a dialog for saving the recently edited palette is presented if any editing was carried out and the active window is updated with the selected palette. The palette files are stored as .LEV files in the application's directory.

## Show palette bar

This command acts as a toggle. When turned on, the palette bar is plotted at the uppermost right corner of the working area. The color used for painting the numbers and bar borders can be changed by selecting the Information item in the Color selection of the Controls area.

## Show electrodes

This command acts as a toggle. When turned on, the electrodes are plotted over the surface map as hemispheres. The color used for electrodes can be changed by selecting the Electrodes item in the Color selection of the Controls area. Notice that when electrodes are shown the application performance is degraded.

## Change mode type   Mode: Color ▾

This command selects the mode type for plotting (maps and tomographic solutions). The Color mode plots the data using the active scale type and palette using seudocolor. The Contour mode plots the data be means of isocontours using the colors given by the active palette. The Contour/Color mode is similar to the Color mode but plots also over the colors the isocontours using the color defined by the Contour/Color item in the Color selection of the Controls area The following figure shows the 3 modes from left to right.
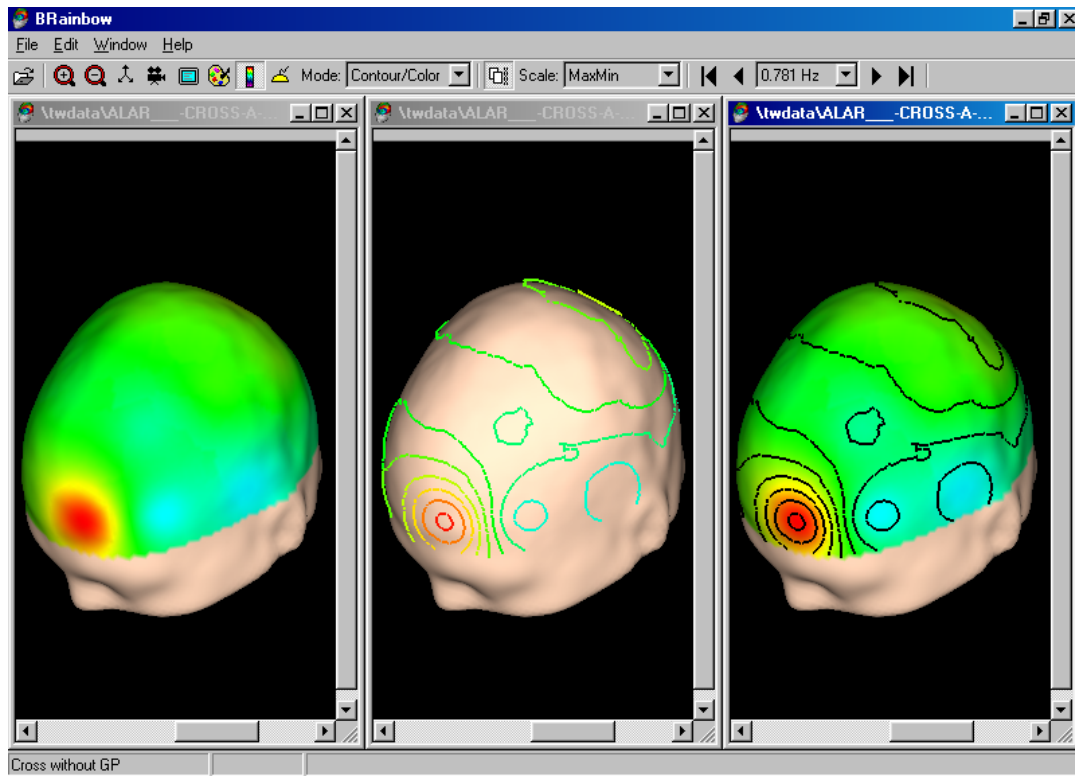
## Yoke windows 🗗

This command acts as a toggle and is enabled if there is more than one data window loaded. When turned on, any surface handling (rotation, zooms, panning) carried out over the active window is carried out also in all the windows loaded. The commands Change scale type, Change mode type, Show/hide electrodes, Show/hide palette bar, Change palette as well as changing the temporal data to plot are affected also by this command.

## Change scale type   Scale: MaxMin ▼

With this command is possible to scale the data loaded in the active window. When activated, a dialog is shown that sets the scale type and its parameters. There are 9 scale types:

**MaxMin**          In this scale type the values minimum and maximum for scaling the data are defined. When this type is selected two edit boxes are shown: Minimum and Maximum for editing the values. The toggle Best Fit automatically adjusts the scale based on the extreme values of the data loaded.

**Threshold**       With this scale type is possible to focus on the positive and negative values in data loaded. When this type is selected the

edit box Threshold is shown to define the threshold that will be used as the scale limit in both directions: positive and negative. The toggle Best Fit automatically adjusts the scale based on the extreme value of the data loaded.

**Step**

This scale type highlights the peak values inside the data. When this type is selected the edit box Step is shown for defining the scaling parameter, which is a positive value. Bigger values cause more highlighted peaks. A value of 0 is equivalent to a Threshold scale with Best Fit turned on.

**Pos/Neg Peak**

This scale type explicitly visualizes the peak positive and negative values inside the data. This effect is achieved by temporarily zeroing all the values in the data that do not coincide with the peak values. When this scale type is selected, the edit box Maximum is shown to define the value that will serve as extreme value in both directions. The toggle Best Fit is also shown which when turned on automatically adjusts the scale based on the extreme value of the data loaded.

**Positive Peak**

This scale type explicitly visualizes the positive peak inside the data. This effect is achieved by temporarily zeroing all the values in the data that do not coincide with the peak value. When this scale type is selected, the edit box Maximum is shown to define the value that will serve as extreme value. The toggle Best Fit is also shown which when turned on automatically adjusts the scale based on the extreme value of the data loaded.

**Percent**

With this scale type it is possible to work with the values of the data in relative rather than real magnitudes. Conceptually it is similar to the MaxMin type but the values are scaled by percent. For the percent conversion the maximum value of the data is taken as 100% converting the rest of the values in relation to the maximum. When this type is selected two edit boxes are shown: Minimum and Maximum for defining the values in % and the toggle Best Fit which when turned on causes an automatic adjustment based on the extreme values of data loaded.

**Peak Percent**

This scale type is similar to Percent but here it is possible to determine the sign of the peak value that will be taken as the reference of 100%. When this type is selected the edit box Minimum is shown to define the minimum value in % (the maximum is always 100) as well as a radio control Peak to set

the sign of the peak value.

**Spot**                With this scale type only the values of data included in the interval defined by the parameters are visualized. When this type is selected the edit boxes Minimum and Maximum are shown to define the interval used for scaling. If the minimum and maximum of the data are given as parameters, then this scale type is equivalent to a MaxMin scale with Best Fit turned on. Contrasting with the Maxmin scale type, the values outside the interval are not plotted.

**Spot bipolar**        This scale is similar to Spot, only the values of the data included in the interval defined by the parameters are visualized. When this type is selected the edit boxes Minimum and Maximum are shown to define the interval used for scaling in the positive direction. The solution values outside the interval (positive and negative directions) are not plotted.

The BIPOLAR palette (see **Change Palette** command**)** is recommended when using the following scale types: Threshold, Step, Pos/Neg Peak and Positive Peak.

If the toggle Freeze scale parameters (present in the scale type dialog) **is** turned on, every time a new temporal data (see **Change temporal data** command) is plotted, it will be scaled using the last scale parameters used.

## First temporal data   ◀|

This command plots the first data of the temporal sequence (see **Open Map** and **Open surface field** commands). The command is disabled if the active window is not a temporal sequence or if the current data is already the first one in the temporal sequence.

## Previous temporal data   ◀

This command plots the data before the current one in the temporal sequence. The command is disabled if the active window is not a temporal sequence or if the current data is the first one in the temporal sequence.

## Change temporal data   `1.953 Hz ▾`

With this command a data can be selected from the temporal data sequence loaded in the active window by means of the combo box.

## Next temporal data ▶

This command plots the data following the current one in the temporal sequence. The command is disabled if the active window is not a temporal sequence or if the current data is the last one in the temporal sequence.

## Last temporal data ▶|

This command plots the last data in the temporal sequence. The command is disabled if the active window is not a temporal sequence or if the current data is already the last one in the temporal sequence.