SPLIT

An Interactive Visualization Tool for Showing Quantum Data

May 3, 2017

1 Compiling

Assuming you are in $HOME/SPLIT_VIS2
Type the following command from the terminal.
> VTK_DIR=<your_location_to_VTK> cmake -H. -Bbuild
> cd build
> make
> make install
> cd ..
> set OMP_NUM_THREADS=<number_of_threads_to _use>
> export OMP_NUM_THREADS=<number_of_threads_to _use>

2 Starting the program

If you'd like to use our default settings, please type "./run" in the terminal assuming you are in $HOME/SPLIT_VIS2 directory. Please note that you might need to modify $HOME/SPLIT_VIS2 SplitTool/config to make sure the data directory is correct.

Otherwise, you need to execute the program by
LD_LIBRARY_PATH=<your_location_to_VTK> ./bin/splitToolkit <CONFIG_FILE>

3 Data format and Preprocessing

3.1 Data format

Two types of data formats are supported in the program:

Type 1:
#line 1: position_x position_y position_z old_region_value value direction_x direction_y direction_z magnitude region_value
...

Type 2:
#line 1: position_x position_y position_z old_region_value value direction_x direction_y direction_z region_value
...

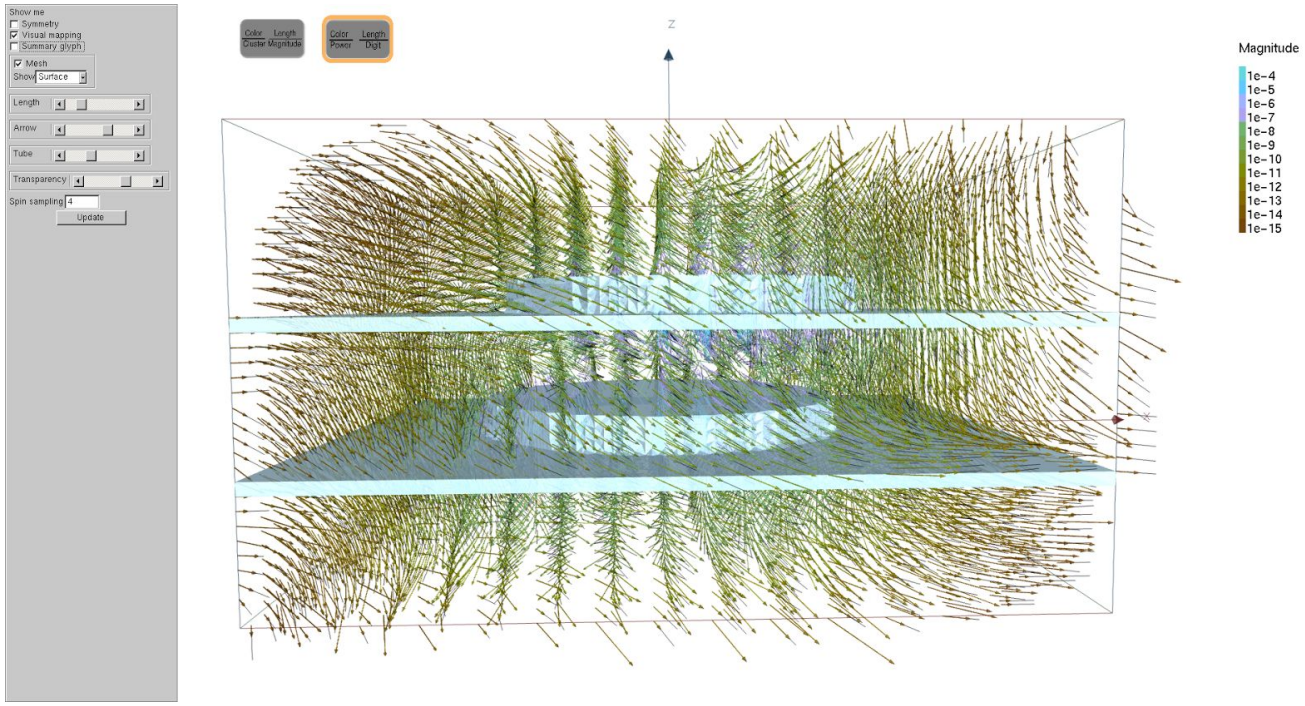
3.2 Input Script

The default config file is $HOME/SPLIT_VIS2/SplitTool/config

#datadir:
<Your directory of data>
#dataname:
<The name of your data>
#folderToStoreFiles:
<The name of a folder to store temporary files>
#planeCenter <position_x position_y position_z>
#planeDirection <direction_x direction_y direction_z>
#planeDistance <distance of two adjacent layers>
#ifcontour:
<0 or 1. 1: contour data. 0: original data>
#contours:
<G AorR value. Here A means absolute value and R means ratio value.>
#clusters:
#step1:
#weights:
#layer: <start_layer, end_layer. For example, if there are 109 layers, the two numbers are 0 and 108>
#bywhole: <1 or 0. 1 means clustering all selected layers together. 0 means clustering each selected layer individually>
#magrange: <min_mag max_mag>
#numofclusters:
#step2:
#weights: <Seven weight values>
#layer: <start_layer, end_layer. For example, if there are 109 layers, the two numbers are 0 and 108>
#bywhole: <1 or 0. 1 means clustering all selected layers together. 0 means clustering each selected layer individually>
#magrange: <min_mag max_mag>
#numofclusters:
#symmetry
#planeCenter <position_x position_y position_z>
#planeDirection <direction_x direction_y direction_z>
#planeX <x y z>
#planeY <x y z>

4. Operations

4.1. Load the SPLIT tool



Zoom in/out: holding the right button
Pan: holding the left button and “shift” button
Rotate: holding the left button
Home button: “h” or “H” button

Modify the input script as the following steps.

Step 1. Define layers (cutting plane)

- Replace *#planeCenter, #planeDirection,#planeDistance* using the information of expected layers in the input script.
- Delete the folder which stores temporary file. For example, if the data name is “z_10z” data and the given name of *#folderToStoreFiles* in your input script is “tmp”, please delete $HOME/SPLIT_VIS2/tmp/z_10z folder.

Please note that the current program only supports one type of layer for each data. So if you want to load the same data multiple times with different types of planes, please delete the folder which stores temporary file before starting the program.

Step 2. Define Parameters of Clustering

The program supports two-step clustering.

Please do as the following instructions to achieve the first step clustering (*#step1* in the input script).

- Define weights. Modify *#weight* in your script file. The seven numbers are weights of position_x, position_y, position_z, direction_x, direction_y, direction_z and magnitude.
- Define regions. Modify *#layer* by giving two numbers: the minimum index of select layers and the maximum index. For example, if there are 109 layers and you expect to see all layers, the two numbers are 0 and 108.
- Modify *#bywhole* in your script file.
- Modify *#magrange* by giving two numbers: the minimum magnitude and maximum magnitude.
- Define # of clusters by modifying *#numofclusters.*

If you want to have the second step clustering, please do the above operations for *#step2.* Otherwise, please give some invalid information in *#step2.* For example, one option is

#step2:
…
#layer: 0 -1
…

Step 3. Define Contour Values

Here is an example

#contours:
G A 1e-6 *//Generate contours on each layer with the value as 1e-6*
or
G R 0.5 *//Generate contours on each layer with the value 0.5 x the maximum magnitude on the layer*

Step 4. Define a Symmetry Plane

- Define the center of one symmetric plane (*planeCenter*)
- Define the direction of one symmetric plane (*planeDirection*)
- Define the two directions on the symmetric plane (*planeX* and *planeY*)

Step 5. Run the program

- LD_LIBRARY_PATH=<your_location_to_VTK> ./bin/splitToolkit <CONFIG_FILE>

4.2 Symmetry

Step 1: To display buttons to control symmetry, please check the “Symmetry” box.
Step 2: Click one or multiple buttons. Now you will see the data satisfies all the selected

symmetric patterns.
Step 3: Click “update”. The clusters will be re-generated based on the selected symmetric patterns.

4.3 Summary glyph

To enable/disable the summary glyphs, please check/uncheck the “Summary glyph” box.

4.4 Visibility

Please press “l” or “L” button to define the visible regions, and input the indexes of layers in the terminal
Here is an example of information displayed in the terminal after “l” or “L” button has been pressed.

Please input the index of the min layer
...
Please input the index of the max layer
...

To toggle each individual layers or multiple layers,

Step 1: Please press “l” or “L” button to define the visible regions, and input the indexes of layers in the terminal.
Step 2: Press “ t” to decrease the indexes of all selected layers (shift all selected layers), or press “T” to increase the indexes of all selected layers.
For example, if the indexes of the selected layers are from 50 to 60, then the new indexes of displayed layers will be 51 to 61 when pressing “T” button.

The locations of displayed layers will be shown in terminal.

4.5 Adjust scaling

To increase/decrease the length of lines, the size of arrows, or the radius of tubes:
Step 1: Please drag the corresponding widgets. Now you can see some sampled data displayed to show your adjustment.
Step 2: Please click “Update” button to apply your adjustment for all datasets.

4.5 Region

To display/hide the 3D boundary of one region, please check/uncheck the “Mesh” box.
Two methods are provided in the program: wireframe and surface.

4.6 Visualization methods

Two visualization methods are provided in the program:
Method 1: Magnitudes are mapped to length of tubes and clusters are mapped to colors.
Method 2: Digits of magnitudes are mapped to length of tubes and powers of magnitudes are mapped to colors.

To toggle the two methods,
Step 1: Please check the “Visual mapping” box.
Step 2: Click one button.
Step 3: Click “Update” button.

4.7 Spin sampling

Step 1: Edit “spin sampling”. It may take 5 seconds to update your input when the previous spin sampling is 1.
Step 2: Click “Update” button.