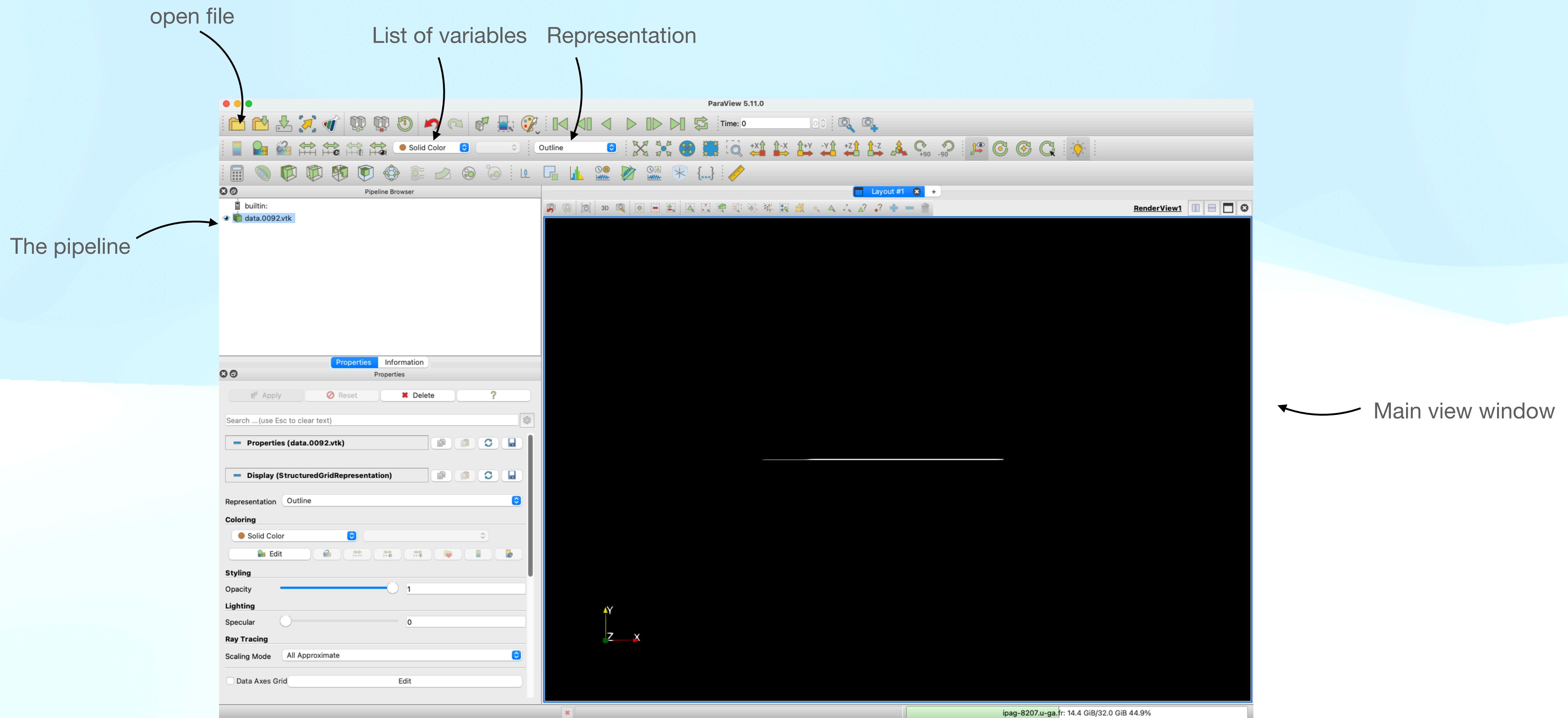


Paraview

An Idefix tutorial

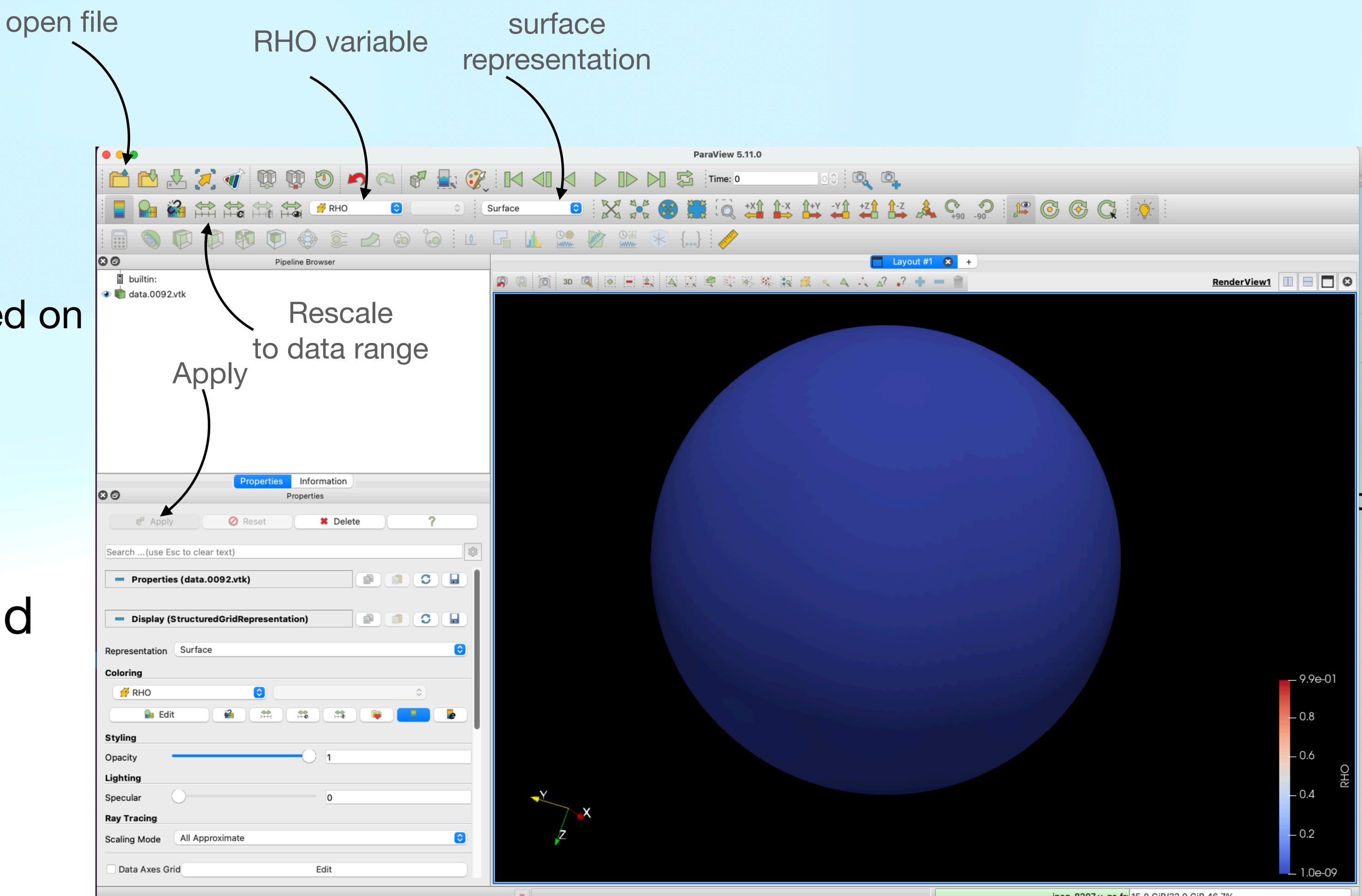
Geoff Lesur, July 5th 2023

Overview



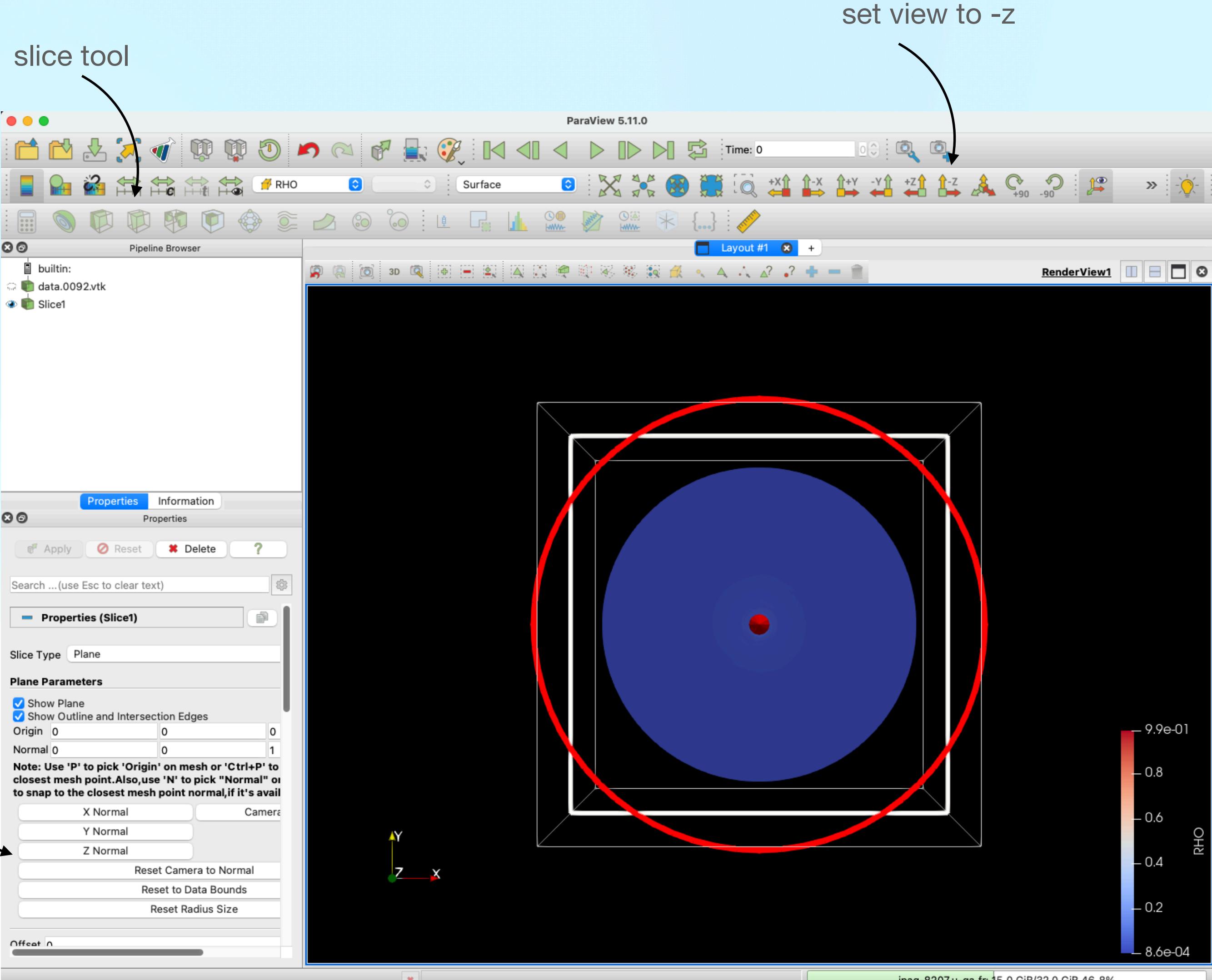
Getting started

- Open the vtk file
(It represents a 3D MRI-turbulent disc computed on Jean-Zay V100 GPUs)
- Click « apply » to load the vtk file content in memory
- Choose a surface representation, and RHO as the variable
- click on « rescale to data range »
- You now see the whole dataset, and essentially the outer radial boundary, which is quite uninteresting...!



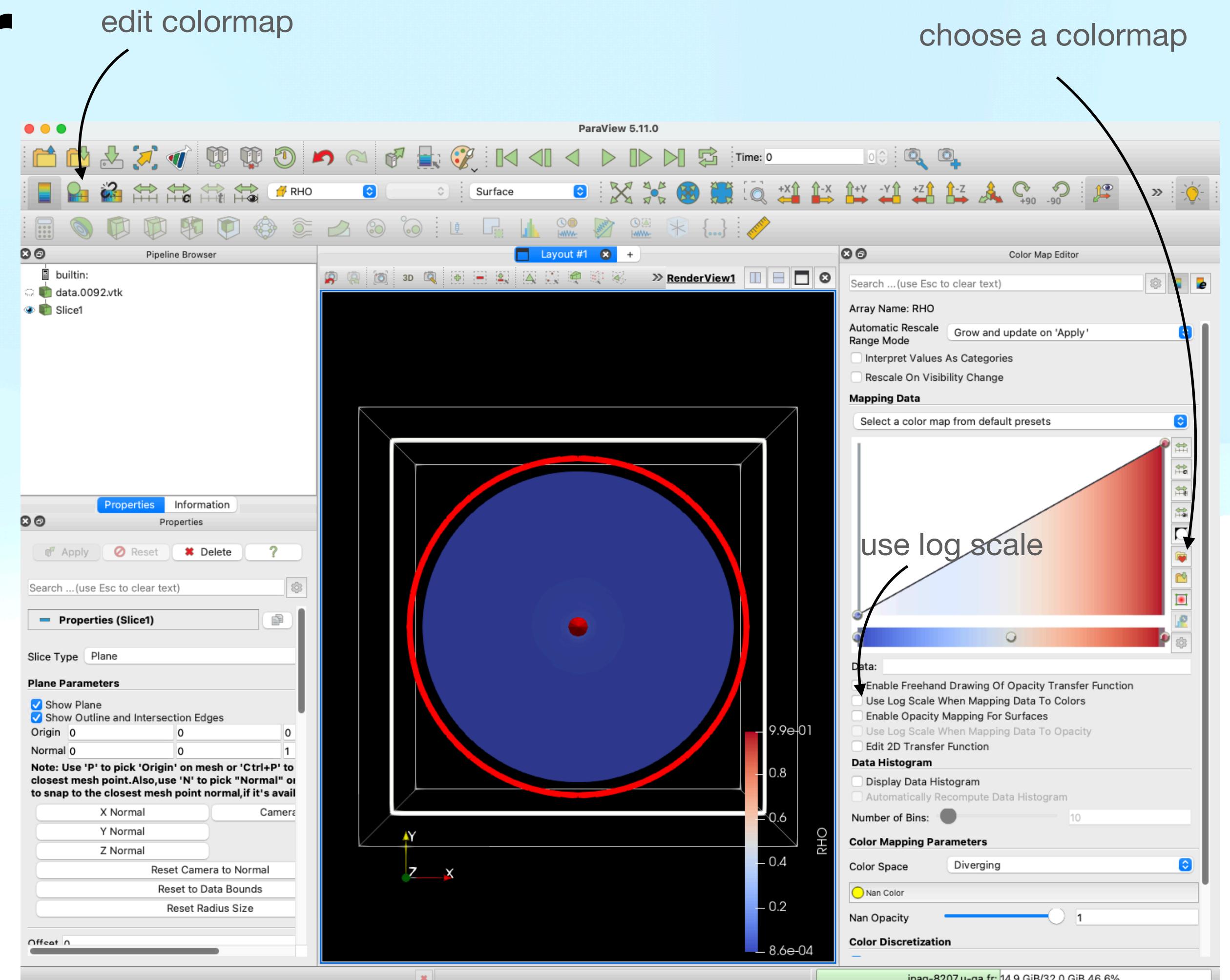
Let's make a cut

- Use the « slice » tool to slice the spherical domain
- Choose « Z normal » to make the slice in the X-Y plane
- click apply
- Reorient the camera so that we look at the domain from the top
- Check that RHO is still the displayed variable



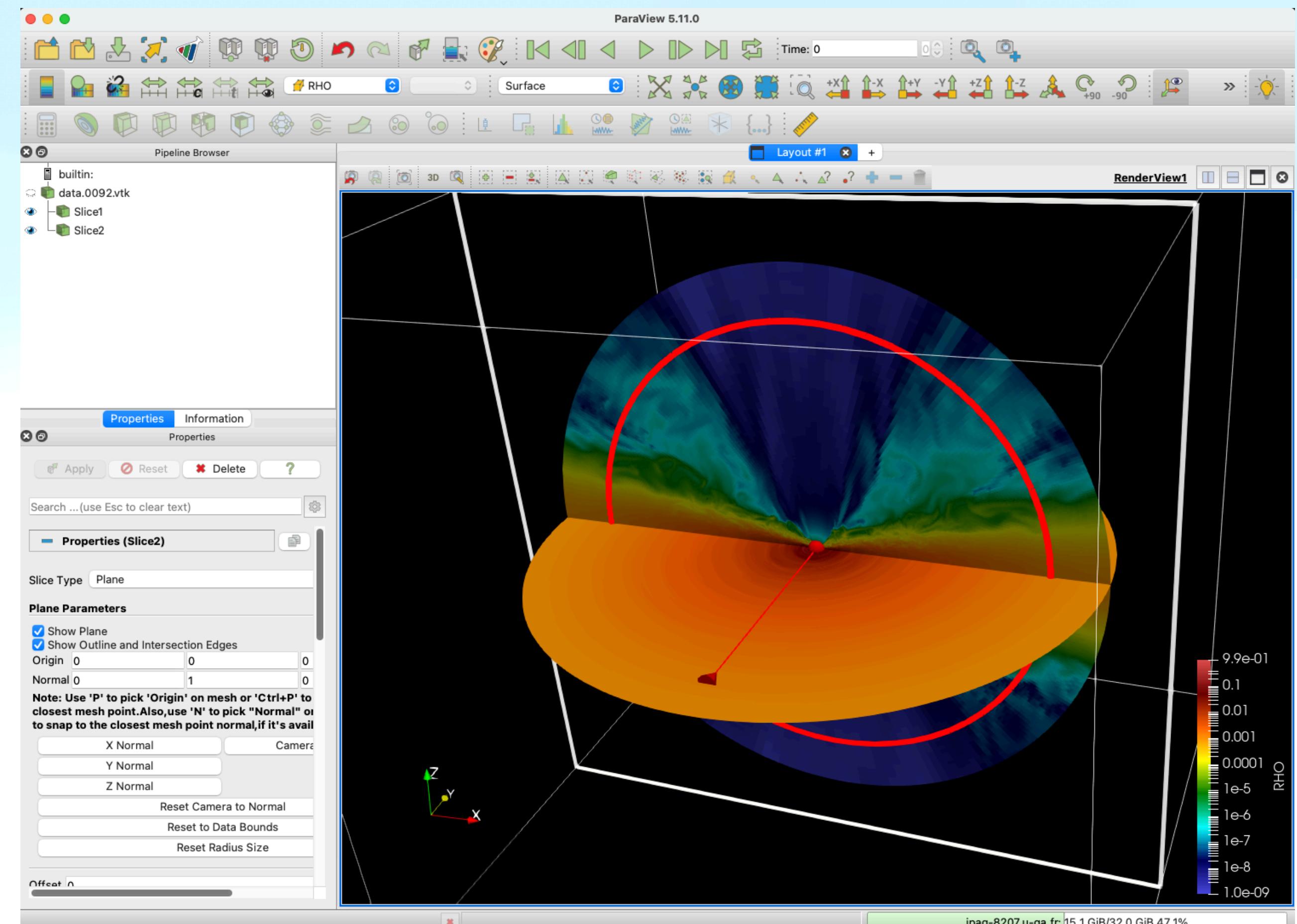
Change the color bar

- Click on « edit colormap »
- Choose « use log scale when mapping data to colors »
- Choose your favorite colormap



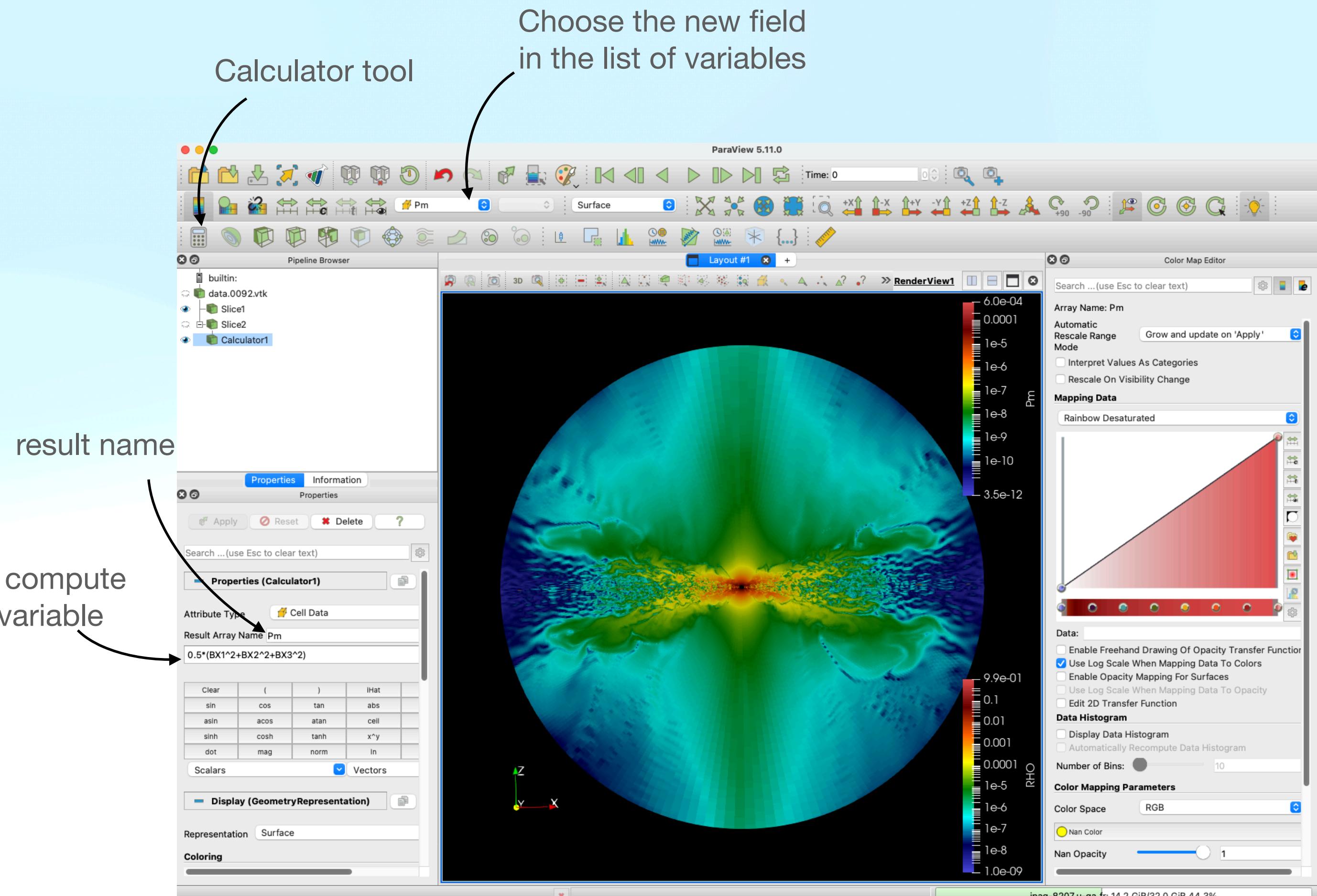
Let's add another cut

- Use the « slice » tool to slice the spherical domain (Warning, you should select the full dataset in the pipeline, otherwise you'll make a slice of a slice!)
- Choose « Y normal » to make the slice in the X-Z plane
- click apply
- Reorient the camera so that we see both slices
- Check that RHO is still the displayed variable



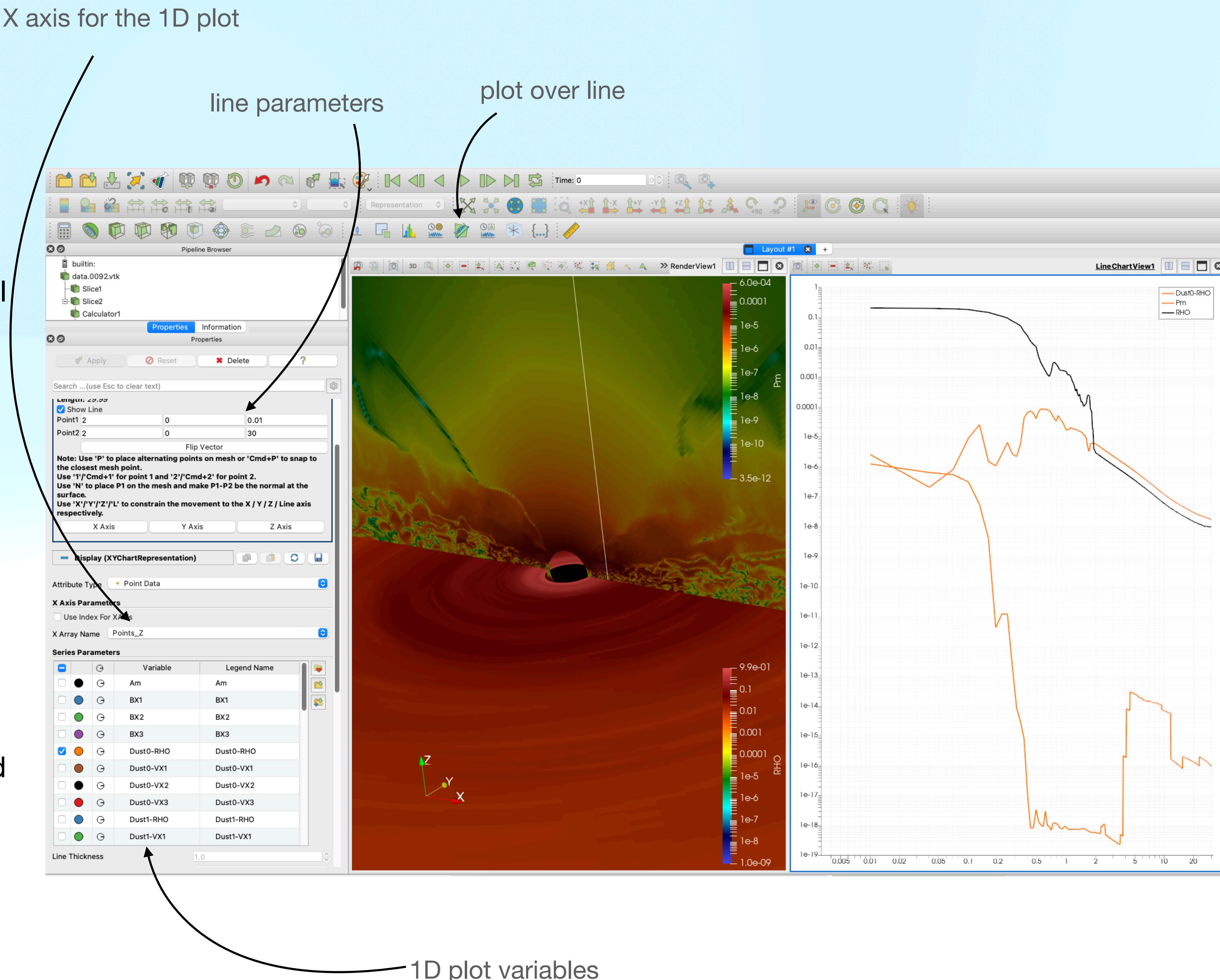
Compute and plot a custom field

- Let's compute the magnetic pressure
$$P_m = \frac{1}{2} (B_r^2 + B_\theta^2 + B_\varphi^2)$$
- Create a calculator taking our slice 2 as an input
- name the result « Pm », and enter the formula:
$$0.5*(BX1^2+BX2^2+BX3^2)$$
- Now choose the newly created « Pm » in the variable list. Open the colormap editor, and use a log scale representation



1D plot

- Let's make a 1D plot of the density and magnetic pressure along z at R=2
- Take the calculator in the pipeline, so that our next filter will have access to « Pm »
- Choose « plot over line »
- Make a line that goes from 2, 0, 0.01 to 2,0, 30 in the « line parameters »
- Apply
- Only plot RHO, Dust0-RHO and Pm (pick up the variables in the « series parameters » of the PlotOverLine properties)
- Specify that we want to use the Z Coordinate as the X axis of the 1D plot
- To plot this in log log, go down the « properties » panel and click « left axis log scale » and « bottom axis log scale »



Streamlines/fieldlines

- Field lines and streamlines can only be computed from point wise vector, while Idefix produces cell-wise scalars
- We must first convert cell data to point data: select the original 3D dataset, then go to « Filters->Alphabetical->cell data to point data »
- From this point, the data is sampled on points. We can therefore use the calculator to construct the magnetic field vector from its spherical components (note that paraview always uses a cartesian coordinate system)

$$B_r \frac{xe_x + ye_y + ze_z}{\sqrt{x^2 + y^2 + z^2}} +$$

$$B_\theta \frac{xze_x + yze_y - (x^2 + y^2)e_z}{\sqrt{(x^2 + y^2 + z^2)(x^2 + y^2)}} +$$

$$B_\varphi \frac{xe_y - ye_x}{\sqrt{x^2 + y^2}}$$

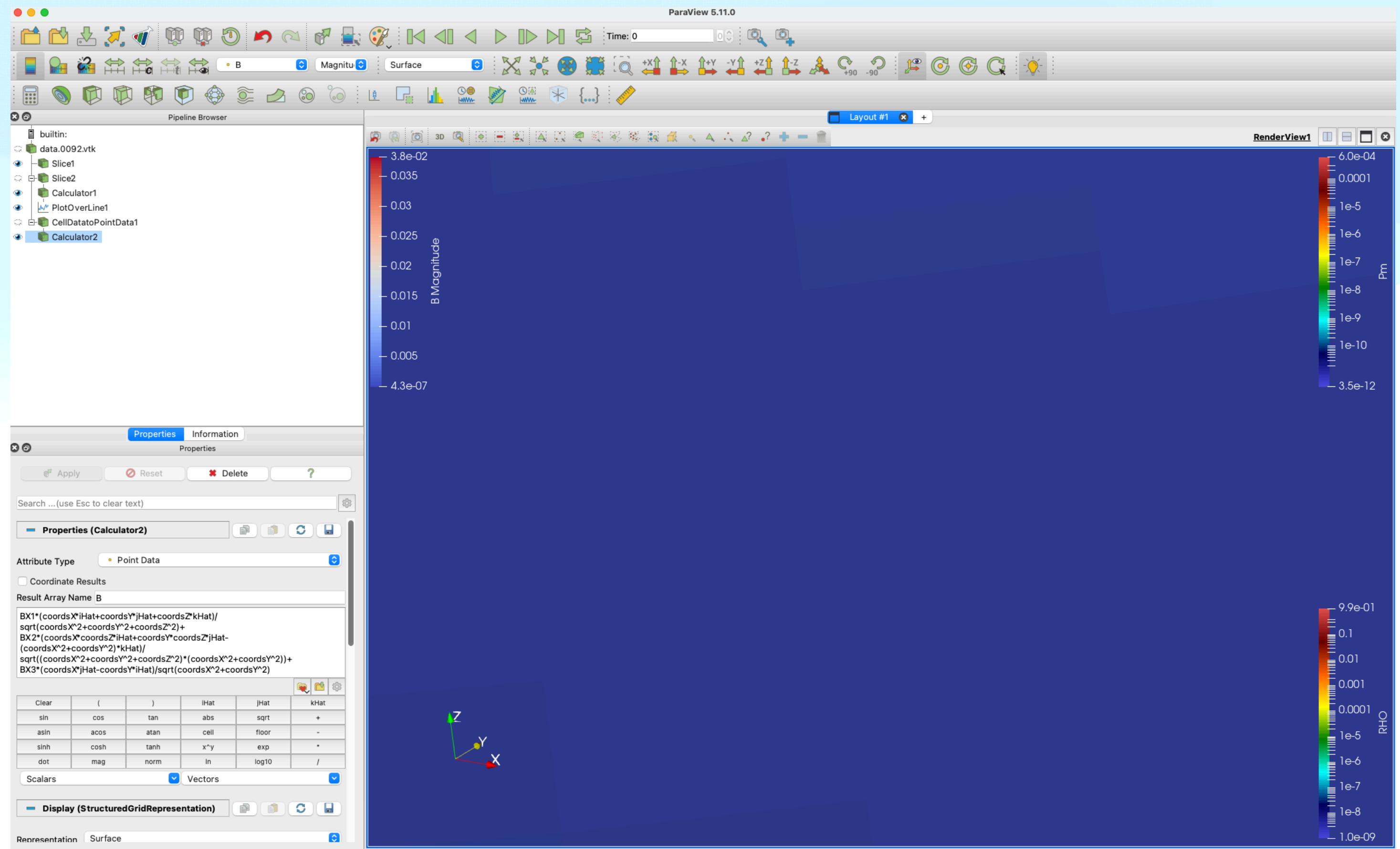
- Which gives in paraview calculator (note that iHAT, jHAT and kHAT are the unit vector e_x , e_y and e_z):

$BX1 * (\text{coordsX} * \text{iHAT} + \text{coordsY} * \text{jHAT} + \text{coordsZ} * \text{kHAT}) / \sqrt{(\text{coordsX}^2 + \text{coordsY}^2 + \text{coordsZ}^2)} +$

$BX2 * (\text{coordsX} * \text{coordsZ} * \text{iHAT} + \text{coordsY} * \text{coordsZ} * \text{jHAT} - (\text{coordsX}^2 + \text{coordsY}^2) * \text{kHAT}) / \sqrt{((\text{coordsX}^2 + \text{coordsY}^2 + \text{coordsZ}^2) * (\text{coordsX}^2 + \text{coordsY}^2))} +$

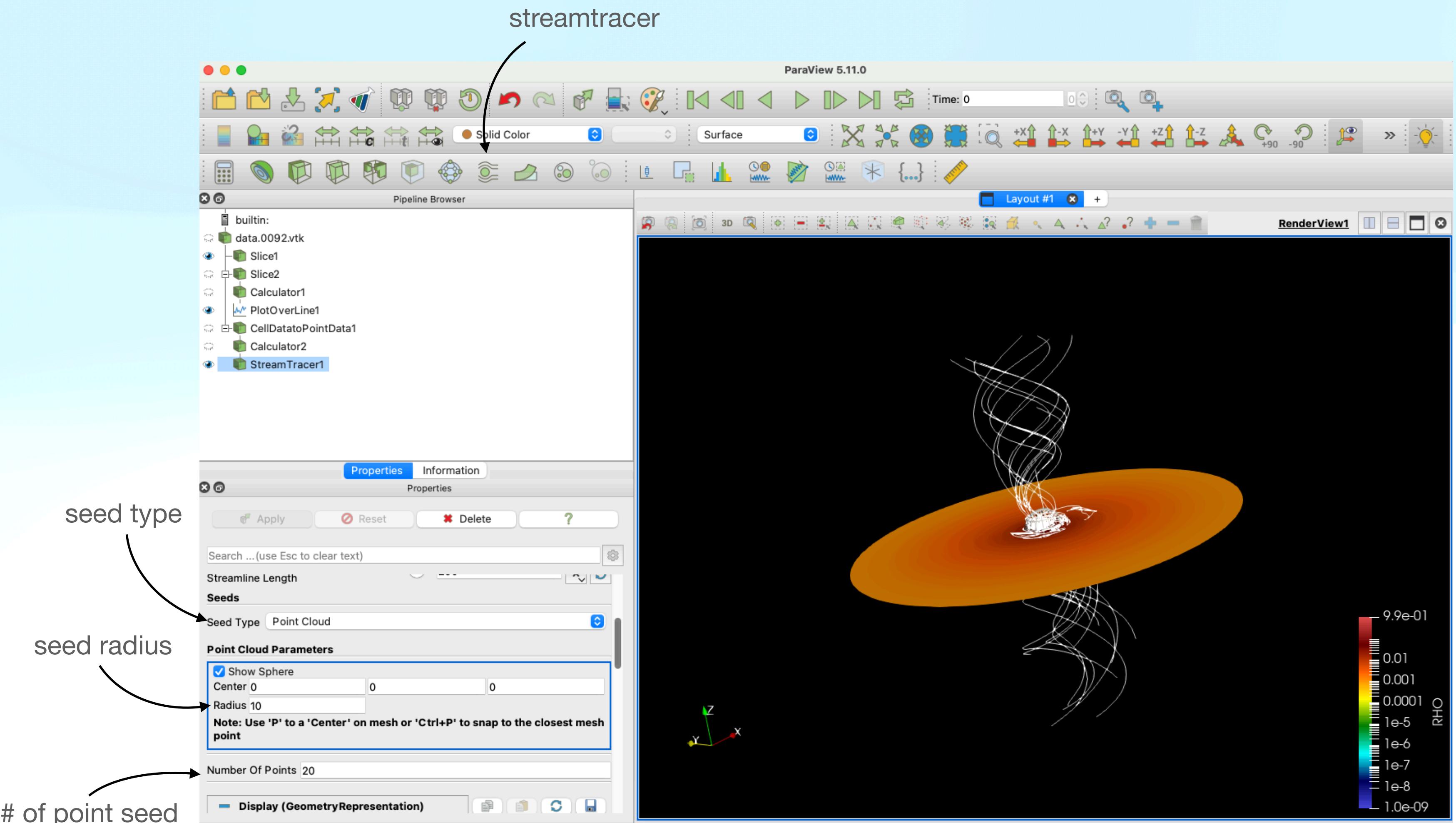
$BX3 * (\text{coordsX} * \text{jHAT} - \text{coordsY} * \text{iHAT}) / \sqrt{(\text{coordsX}^2 + \text{coordsY}^2)}$

- Let's name « B » the name of the result of the calculator and apply
- From this point, we have a vector « B », for which we can plot the magnitude and components (check the variable list)



Streamlines/fieldlines (cont'd)

- With the vector B properly defined, we can plot the field lines, click on « streamtracer »
- The streamtracer uses a numerical integration to represents the streamlines, hence one need to give it a set of starting points: the « seeds »
- Let's choose a "point cloud" of radius 10, and only 20 seed points. Paraview will pick up random points at the sphere surface as starting points.
- Hide the second slice (click on the « eye » in the pipeline) to view your field line
- Extra: to give it a little more perspective, make the field line like tubes: go to « filters->alphabetical->tube » Feel free to color the field lines by whatever variable you want



Volume rendering

- Volume rendering is very time-consuming on an unstructured grid (checkout the information tab). Hence, it is recommended to re-mesh onto a regular cartesian grid before attempting any volume rendering.
- First pick the « celldatato pointdata » object in your pipeline: this will be our starting point. We are going to remesh its output.
- Go to filters->alphabetical->Resample to image.
- We want to represent the disc with a volume rendering of the density. So we are going to re-mesh only the disc:
 - uncheck « use input bound »
 - use a sampling dimension of 200x200x50 (that's the resolution of the new mesh in x,y,z)
 - for the sampling bound, let's zoom on a box centered in +/-20 in x and y and +/- 2 in z
 - Don't forget to apply!
- Now that we have a uniform rectilinear grid, let's use the volume representation
- Most of the time with volume rendering is spent on finding the right opacity table and colormap range. Here, it is recommended to set the colormap range to [0.01,0.05]
- It is recommended to switch off the field lines (« tube »), or reduce the number of seeds to make a cleaner representation

