

# ParaView

## (User Manual)

### For Geophysical modeling and Inversion

(Last updated: January 2026)

Download and install ParaView from: <https://www.paraview.org>

**1-Open the software.**

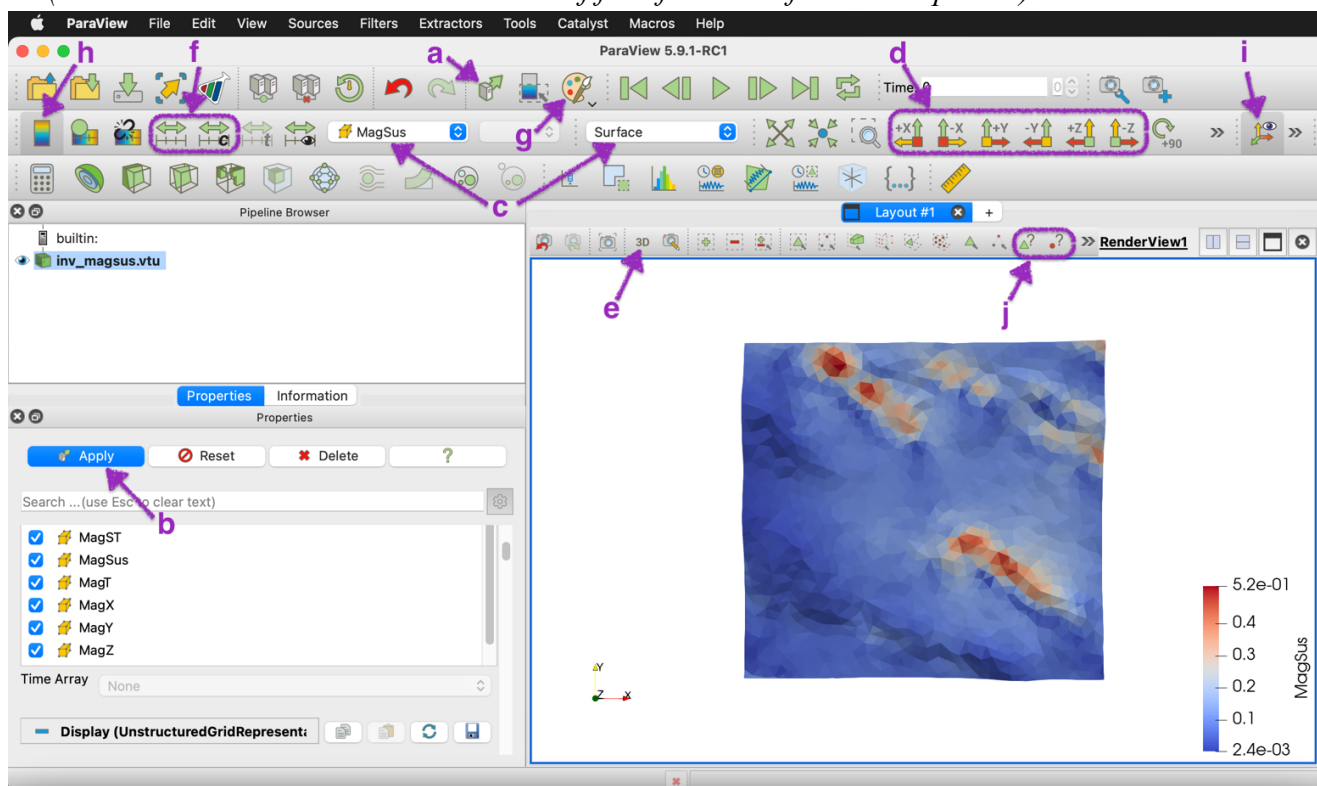
**2-Open your files (vtu, vtk or vtr) by clicking on them, or drag and drop them, or go to menu 'File' => 'Open'. (Open the files one by one.)**

**3-Changes to parameters can be applied automatically by selecting button "a" (in figure below) or manually by clicking button "b" (button 'Apply'). I prefer applying changes manually by clicking button 'Apply', so I deselect button "a".**

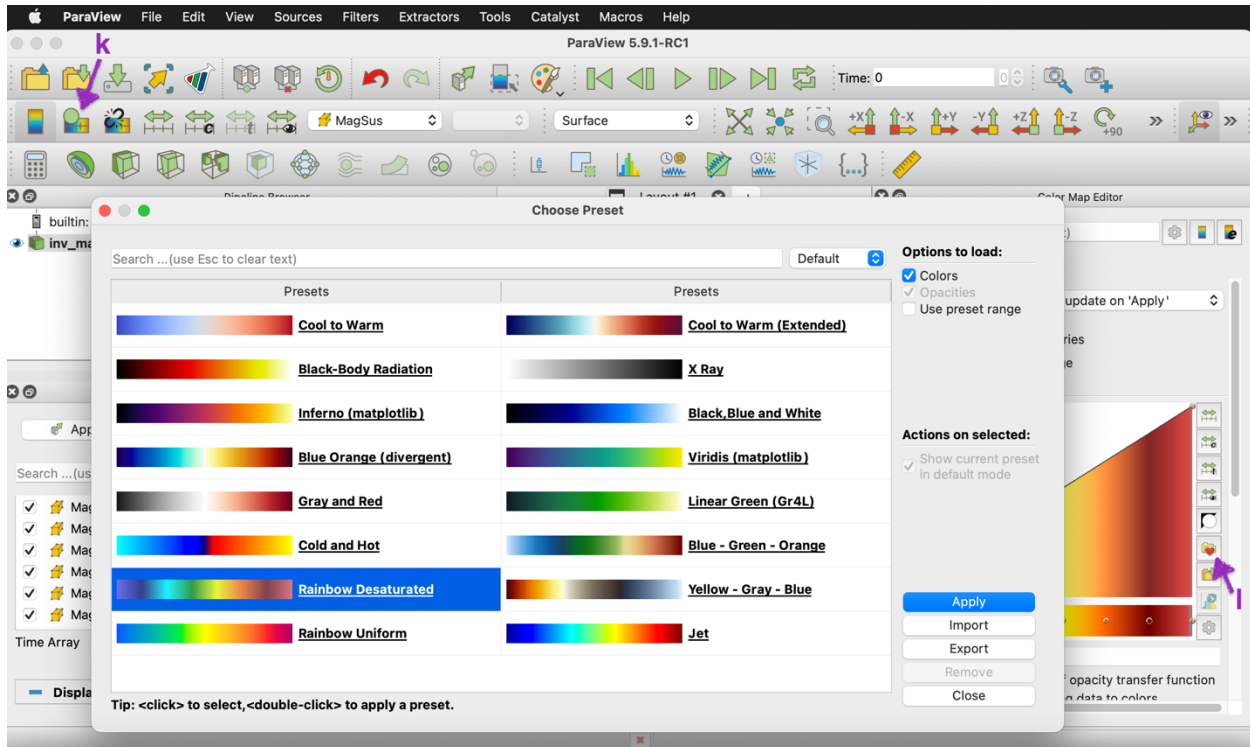
After opening your file, make sure you are selecting appropriate parameters in buttons "c".  
*(Note that after any changes, you should always hit 'Apply' button to see them.)*

**4-You can use your mouse to see your model from different angels, or you can use buttons "d".**  
 You choose 2D and/or 3D modes by clicking button "e". In 2D mode, you will not be able to rotate your model anymore. Coordinates 'Y' and 'X' are usually northing and easting, respectively.

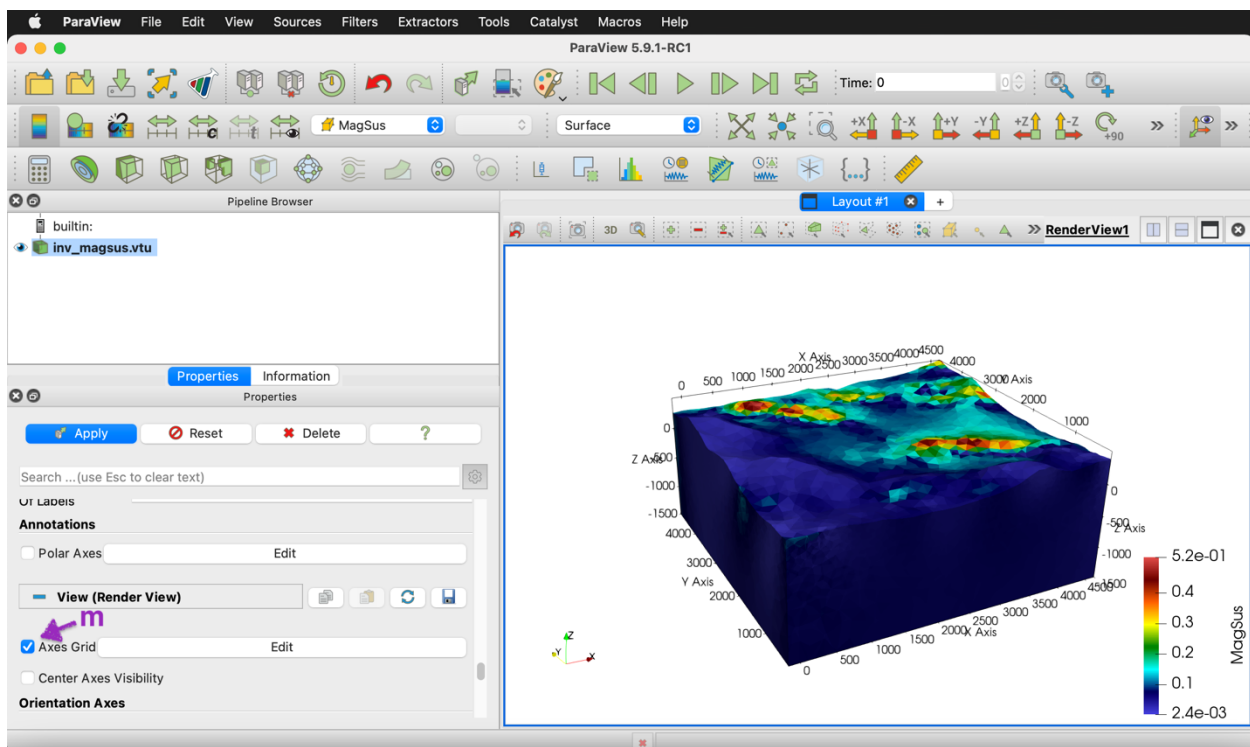
**5-You can rescale the data range by buttons "f" (make sure you always rescale your model after any changes by clicking on the first button from the left), and change the background color by button "g".** The color legend bar and direction arrows can be hidden by "h" and "i", respectively.  
 You can find the node and cell information (e.g., coordinates, cell values,...) using buttons "j".  
*(Note that each tetrahedral cell is made of four faces and four nodes/points.)*



6-To change the color used in the model, you should first click button “k”. Then a panel (Color Map Editor) will be opened on the right side at which you can click button “I” to see the options (I usually use “Rainbow Desaturated” and “Jet” colors). In this panel, you can invert the colormap and/or switch to the log scale as well. Also, try different options for “Color Space”, “Data Histogram”, and “Color Discretization”.

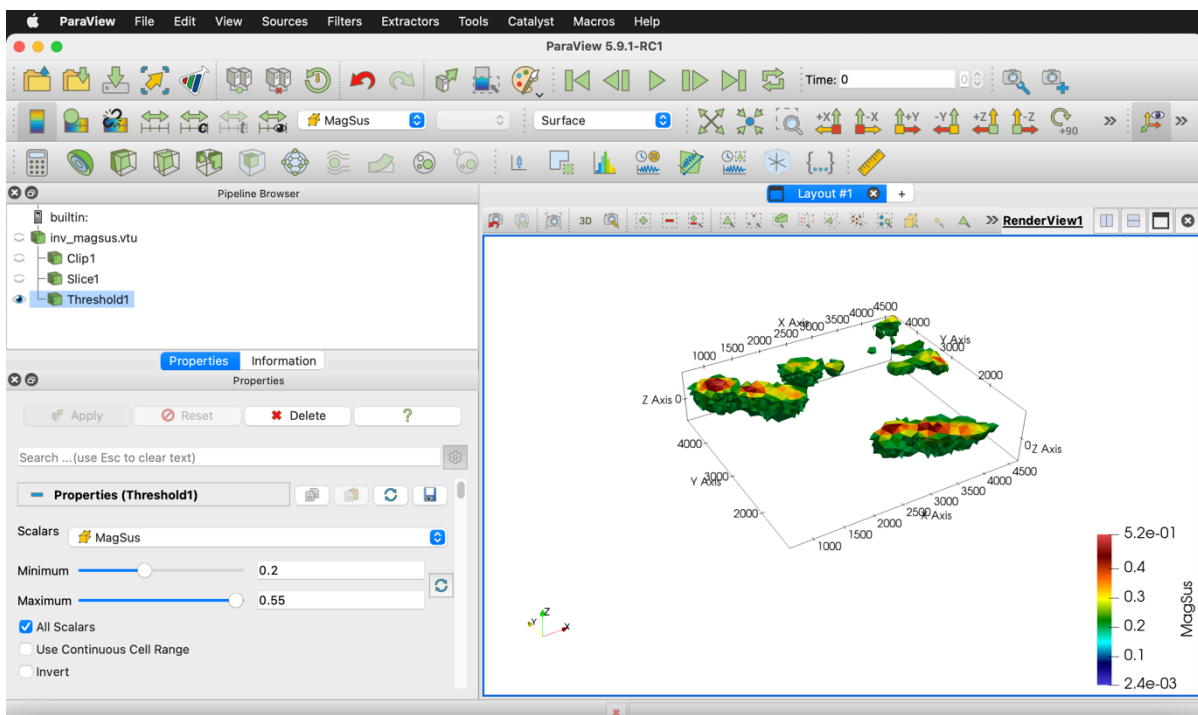
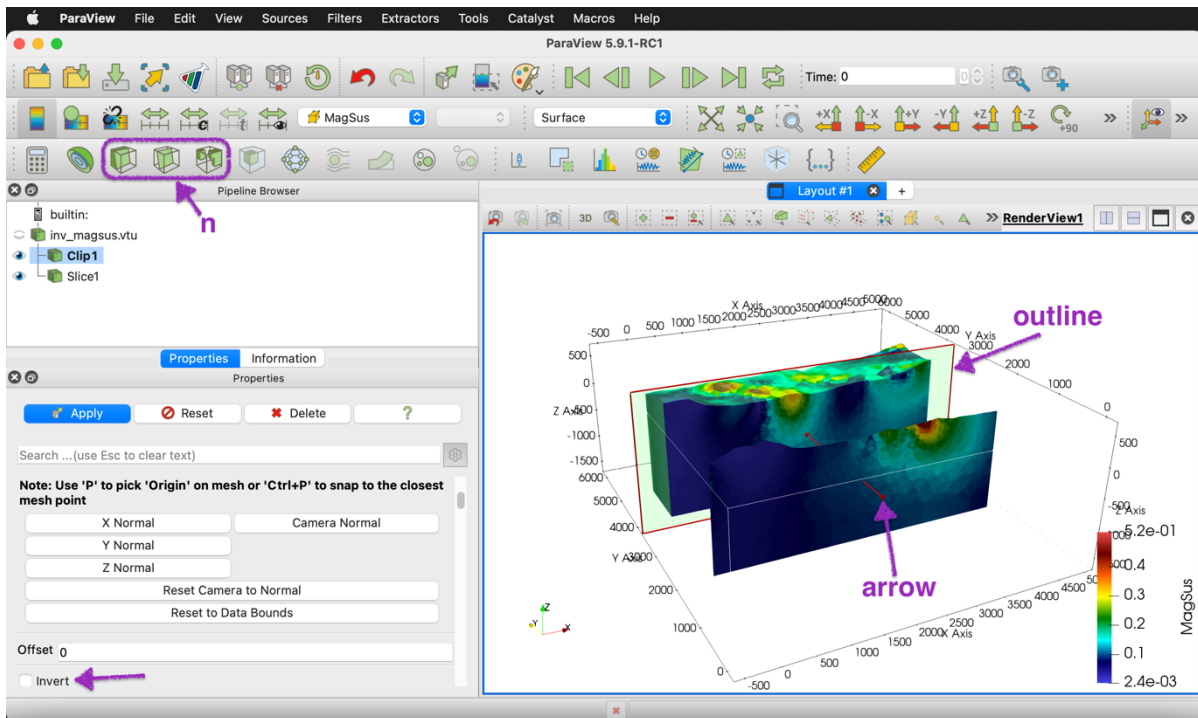


7-To see axes, in the ‘Properties’ panel (on the left side), under the ‘View’ section, check the ‘Axes Grid’ checkbox (button ‘m’).



**8-**To create Slice, Threshold or Clip, click one of the buttons “n”. Next, in the ‘Properties’ panel and under ‘Plane Parameters’ section, you can change your Slice or Clip options. You can also easily do it by moving the red outline and the arrow by mouse. For Clip, you might need to use ‘Invert’ and ‘Crinkle’ checkboxes as well. For Threshold, under the ‘Properties’ section, initially select an appropriate option for ‘Scalar’, and then define ‘Minimum’ and ‘Maximum’ values.

*(Make sure that after any changes, you are selecting appropriate parameters in buttons “c” as well.)*



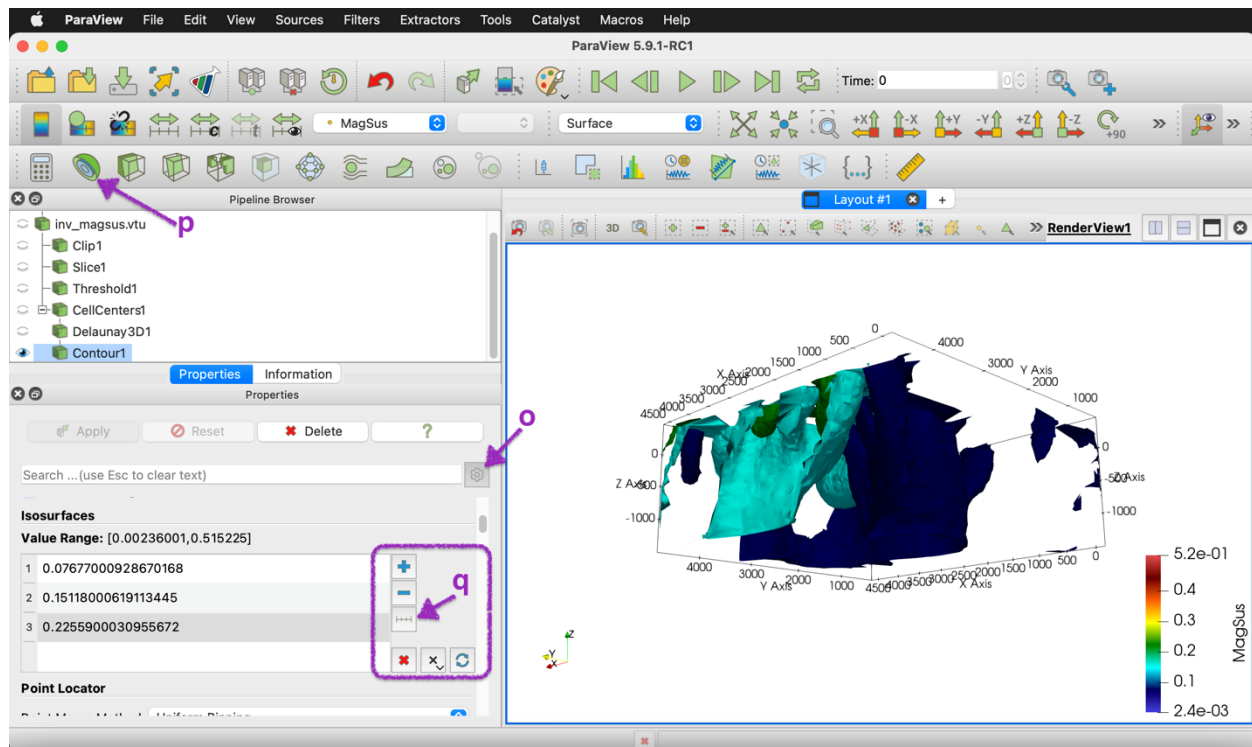
**9-**In the ‘Properties’ panel, you can change the opacity and node and line sizes in ‘Styling’ section. You can also transform your model by section ‘Transforming’. Note that, you can find them only if you hit the button “o” (see figure below).

**10-**You can generate nodes at the centre of cells by ‘Cell Centers’ (from menu ‘Filter’ => ‘Alphabetical’). Cell values will be given to the nodes. Do not forget to select ‘Vertex Cells’ from ‘Properties’ panel.

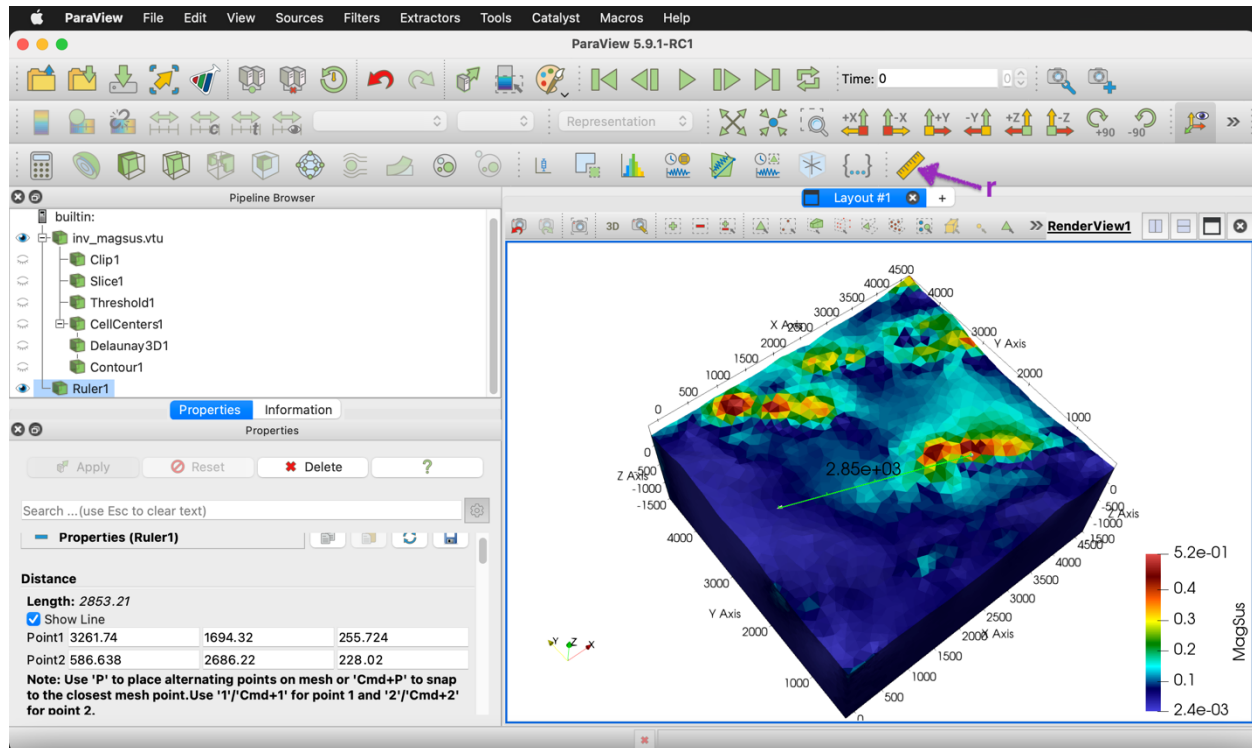
**11-**To generate triangular surfaces from your nodes/points, apply ‘Delaunay 2D’ (from menu ‘Filter’ => ‘Alphabetical’) to your nodes/points. To generate contours and isosurfaces from nodes, initially apply ‘Delaunay 2D’ and ‘Delaunay 3D’ (from menu ‘Filter’ => ‘Alphabetical’) to your nodes, respectively. You might need to give an appropriate value to “alpha” which is the maximum length/size of cells (e.g., triangles).

Finally, hit ‘Contours’ button (button ‘p’). In the ‘Properties’ panel, select an appropriate option for ‘Contour By’. Also, from section ‘Isosurfaces’ click button “q” to define the contour/isosurface sizes.

To generate smooth isosurfaces from a model, you can also use the method mentioned in Appendix A.



**12-** To measure distance between two points, you can use ‘Ruler’ (button ‘r’). First, use your mouse to point to the first location in your data/model and press the **P** key in your keyboard. Then, use your mouse to point to the second location in your data/model and press **P** key again. Now, hit the ‘Apply’ button.



**13-** You can change light in the model from menu ‘View’ => ‘Light Inspector’

**14-** To generate vectors, you need to use ‘Calculator’ (button “s”) and ‘Glyph’ (button “t”) as follows:

You should first use the ‘Calculator’ button to create a vector field from your three scalar fields. First make sure you are choosing an appropriate option from checkbox ‘Attribute Type’ (in my example, it was ‘Cell Data’). Then, you can name your output file in the ‘Result Array Name’ text field.

Say your three scalar fields are named MagX, MagY, and MagZ. Then, in the main text field just above the various functions/operators (under the ‘Result Array Name’), you should enter:

$$(i\hat{H}at*MagX)+(j\hat{H}at*MagY)+(k\hat{H}at*MagZ)$$

Then use ‘Glyph’ button to map vectors. In the ‘Properties’ panel,

in ‘Glyph Source’, at ‘Glyph Type’ section choose Arrow.

in ‘Orientation’, at ‘Orientation Array’ section choose your ‘Result Array Name’.

in ‘Scale’, at ‘Scale Array’ section choose an appropriate option.

in ‘Masking’, at ‘Glyph’ Mode section choose one of the options.

in ‘Glyph Transform’, at ‘Scale’ section change the values to change the size of the arrows.

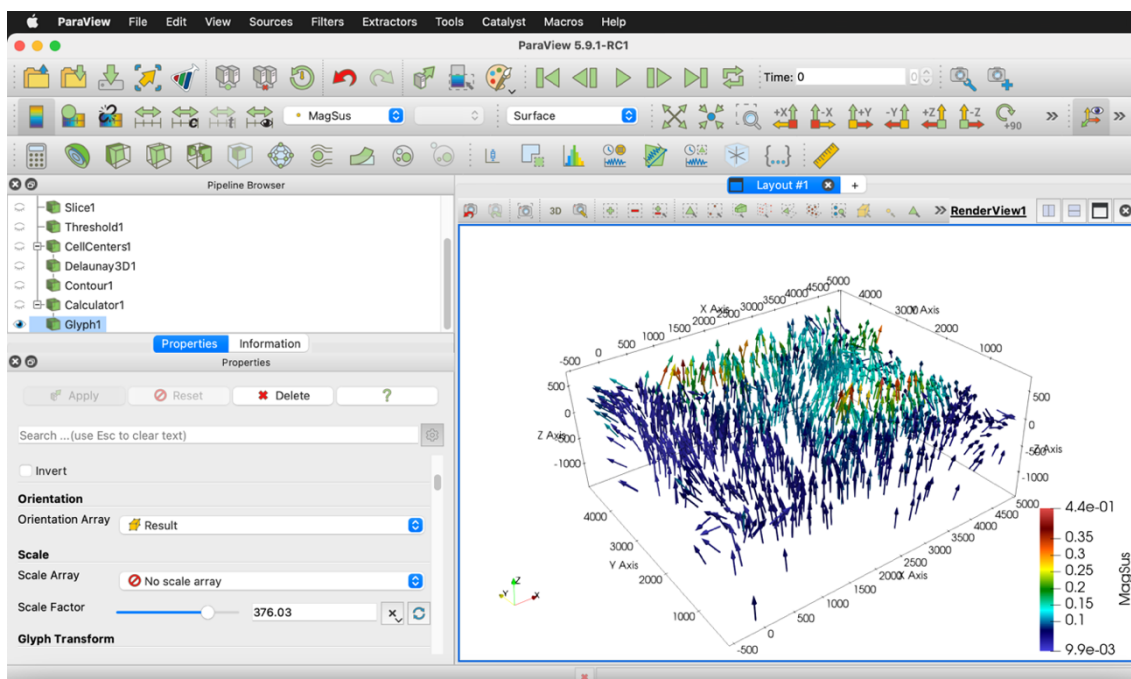
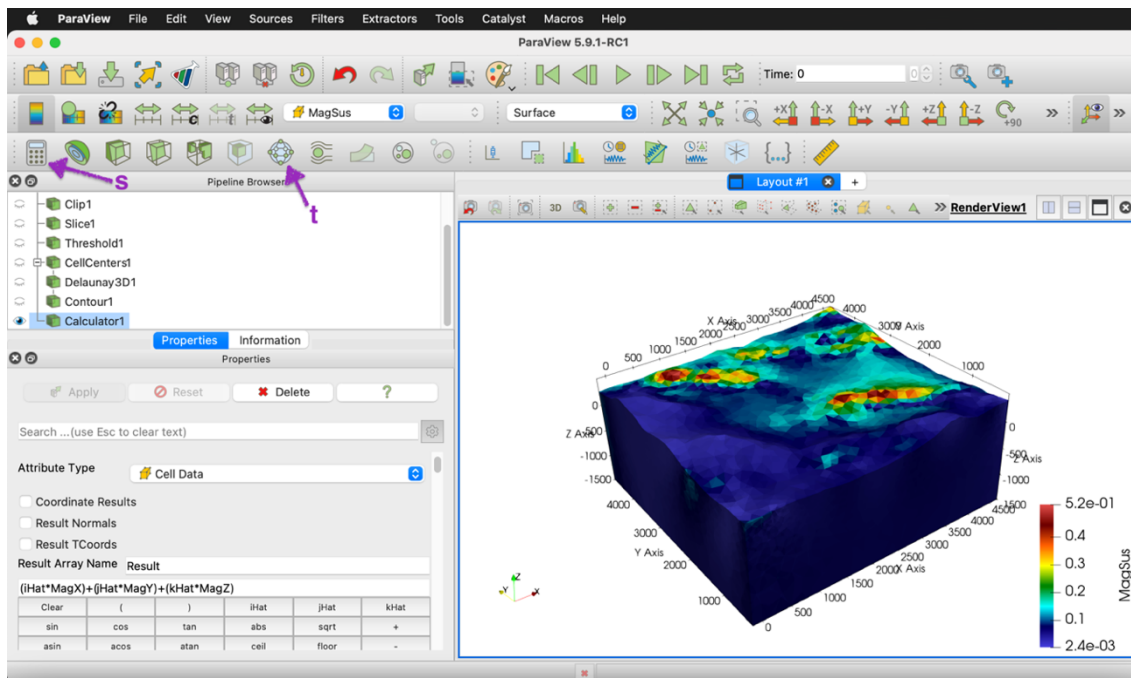


Note that if your three scalar fields of magnetic components are in a standard geophysical system for magnetic data (with +x North, +y East, and +z down), then you should enter the following command to calculate magnetic vectors in a standard Cartesian system (with +x East, +y North, and +z up):

$$(iHat*MagY)+(jHat*MagX)+(kHat*MagZ)*(-1)$$

Note that if your file already has an option for cell vectors, no need to use ‘Calculator’, and you can only use ‘Glyph’. Just make sure at ‘Orientation Array’ section you choose your cell vector’s option.

*(Do not forget to always hit the ‘Apply’ button to make changes.)*



## APPENDIX A

### Export a point cloud (cell centre values) to .csv/.txt file

As mentioned before, you can generate nodes at the centre of cells by 'Cell Centers', and give the cell values to the nodes. You can export this node cloud using "File>Save Data". Make sure you are giving a value of "10" to "Precision". Note that in the output file, the first column has the cell values, and the next three columns are x, y and z coordinates.

### Import points from .csv file

You can open a ".csv" file by "drag and drop" or from "File>Open". The points in the file will be shown in a table. You can plot them using "Table to Points" at "Filters>Alphabetical". Make sure from the 'Properties' panel (on the left side), you select appropriate columns for 'X Column', 'Y Column', and 'Z Column'.

### Find Data (nodes or cells)

If you have an ID or the number of cells/nodes and want to locate it in your model, use 'Find Data' from the 'View' menu.

### Make geometry shapes

You can generate geometric shapes from 'Sources' → 'Geometry Shapes'. To move, rotate, or deform these shapes, apply the 'Transform' filter from 'Filters' → 'Alphabetical'. (If the surfaces are not triangulated, you might need to apply the 'Triangulate' filter.)

### New Colormaps

The "Colormaps" folder contains several new custom color maps that can be imported into ParaView.

To import these color maps, open the "Color Map Editor" panel as described earlier (then click the button "I" to view the options). In the "Choose Preset" window, click "Import" to import our color maps. To locate the imported color maps, either search for them by name or select "All" instead of "Default" in the preset filter.

### Some useful filters

(From menu 'Filters' → 'Alphabetical')

**Extract Surface:** It converts 3D volumetric model into a 2D skin. It extracts a surface from a body (for example, after applying a Threshold filter to your model).

**Smooth:** It relaxes the points to make triangles more equilateral (even) and removes noise. For parameter “Number of Iterations”, starts with 20. Too many iterations will cause your model to shrink. For parameter “Convergence”, set it to 0 to force it to run the full number of iterations.

**Decimate:** It reduces the triangle count while trying to keep the shape. Use this when your file is too heavy or has too many tiny triangles on flat faces. For parameter “Target Reduction”, a value of 0.9 means it will try to remove 90% of the triangles. For parameter “Preserve Topology”, keep this checked to prevent the filter from tearing holes in your mesh.

**Clean to Grid:** To merge points within a certain distance of each other for deleting sliver triangles. The parameter "Tolerance" is a percentage of the overall size of your model. This is safer if you are not sure of your model's exact measurements. If you are, check "Tolerance Is Absolute" and put the value in "Absolute Tolerance".

**Delaunay 2D:** It creates a brand-new surface from a cloud of points. It only works on a flat plane. You can use it to make a simple data map from your data points. The parameters “Alpha” controls how far the filter will reach to connect points. A small Alpha prevents it from filling in natural holes in your model.

**Triangulate:** This is to ensure every single face has exactly 3 sides! Use this if you have "Polygons" or "Quads" that are causing errors in subdivision or smoothing.

**Transform:** This moves, rotates, or scales your entire model in 3D space.

**Histogram:** A diagnostic tool to see the distribution of your data values. For parameter “Select Input Array”, choose the data you want to graph. For parameter “Bin Count”, the higher numbers give you more detail on the "spread" of your data.

#### NOTES:

Some of the filters above only work on a .vtu file of “Polygonal Mesh” (PolyData) and not an “Unstructured Grid”. Check this out at the 'Information' tab. If "Type" says Unstructured Grid, you need to apply the 'Extract Surface' filter to it to change it to Polygonal Mesh. (Note that, the output of filter “Clean to Grid” is always an Unstructured Grid.)

Also, if you intend to save your model in formats such as .obj or .stl, the model must first be converted to a “Polygonal Mesh”. If the data is in another format (such as Unstructured Grid), these export options will not appear in the 'Save Data' menu.



## New Plugins

Folder 'ParaView\_Plugins' have followings:

- *GeoTIFFExportPlugin.py*: Exports ParaView slices to GeoTIFF format with georeferencing support.
- *GXFExportPlugin.py*: Exports ParaView slices to GXF format.
- *IrregularSlicesPlugin.py*: Create vertical slices along irregular profiles

Loading Plugins in ParaView:

1. Go to 'Tools' → 'Manage Plugins...'
2. Click 'Load New...'
3. Navigate to the directory containing the plugin files and select the desired plugin file (`.py`)
4. Click 'OK' to load the plugin. The plugin will appear in the list of loaded plugins
5. Expand the plugin entry to select 'Auto Load' checkbox to automatically load the plugin every time ParaView starts
6. Close and restart ParaView

Note: Ensure the plugin status shows "Loaded" to enable it for the current session. Also, once you load a plugin, do not move or rename the directory containing the plugin files, as this will cause the plugins to stop working.

*GeoTIFFExportPlugin.py* and *GXFExportPlugin.py*: Exports ParaView slices to GeoTIFF and GXF formats. After applying these option, they can be seen in menu 'File' → Save Data...'

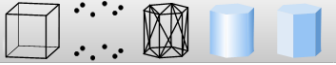

*IrregularSlicesPlugin.py*: This filter creates vertical slices along irregular lines or profiles. The stations along the profiles should be provided in a .vtu file. All stations belonging to the same profile must share a common attribute to distinguish them from other profiles. If no attribute is provided, the software assumes the file contains a single, continuous profile. For this:

- Load your model dataset and your profile dataset (vtu files).
- Select the model in Pipeline Browser.
- Select Irregular Slice filter.
- In Change Input Dialog:
  - Check 'Input', and then click on your model in the list
  - Check 'Profile', and then click on your profile in the list
- In filter Properties:
  - Profile Group Attribute = array that identifies profile IDs.
  - Leave empty if all stations are one profile.
  - Optional: set Profile Station Order Array if you want explicit station ordering; otherwise auto-order is used.
- Click Apply.

## APPENDIX A

### MeshLab

In MeshLab, you can decimate a mesh (i.e., a triangulated surface) while keeping the surface shape, with more equilateral triangles, and avoiding sliver (“silver”) triangles. For this, there are four different methods recommended below, but first consider these:

- Download and install MeshLab from: <https://www.meshlab.net>
- Drag and drop your mesh/surface file (e.g., .obj file) into MeshLab to open it.
- Icons  are for controlling mesh visualization. For example, click on  to see the triangle edges of your mesh.
- Undo and Redo are not straight forward in MeshLab. But, you can duplicate the layer before applying filters so you always have the original layer. For this, right click on it and select “*Duplicate Current Layer*”.
- In MeshLab, to delete your mesh, right click on it and select “*Delete Current Mesh*”.
- To export the files: from menu 'File' → 'Export Mesh As' (e.g., choose '.obj' format, and then select 'None')

#### Method 1: Merge Close Vertices

*Filters → Cleaning and Repairing → Merge Close Vertices*

This merges vertices closer than a given distance. This useful tool merges nodes that are too close, remove near-duplicate vertices, and collapse tiny sliver regions.

Set these parameters:

**Merging distance:** choose based on your model scale (Here, "world unit" represent the absolute physical dimensions of your model (like meters), whereas "perc on" defines a value relative to the total scale of the entire object.)

#### Method 2: Isotropic Explicit Remeshing

*Filters → Remeshing, Simplification and Reconstruction → Remeshing: Isotropic Explicit Remeshing*

This rebuilds triangles evenly, makes triangles more uniform and equilateral, and removes slivers. This is the best method to get near-equilateral triangles.

Set these parameters:

**Iterations:** 5–10

**Adaptive remeshing:** ON

**Target Length:** choose based on your model scale (e.g., try “world unit”)

#### Method 3: Clustering Decimation

*Filters → Remeshing, Simplification and Reconstruction → Simplification: Clustering Decimation*

Fast and but less accurate at preserving exact shape.

Set these parameters:

**Cell Size:** choose based on your model scale (e.g., try “world unit”)

#### **Method 4: Quadric Edge Collapse Decimation**

*Filters → Remeshing, Simplification and Reconstruction → Quadric Edge Collapse Decimation*

Set these parameters carefully:

**Target number of faces or Percentage reduction**

→ Choose how much to reduce (example: 50%)

**Preserve Normal** → ON

Keeps the surface shape accurate.

**Preserve Topology** → ON

Prevents holes and topology damage.

**Optimal Position of Simplified Vertices** → ON

Improves triangle quality and shape.

**Planar Simplification** → ON (important for smooth areas)

**Quality Threshold** → set to 0.3 – 0.6 (recommended 0.5)

Critical settings to avoid sliver triangles (Higher value = fewer skinny triangles)

**Preserve Boundary of the mesh** → ON

Only if needed, otherwise OFF improves triangle regularity.

**Weight Normal** → 0.1–0.3 (optional)

Helps preserve curvature.

Depending on your mesh, use any of the methods above or even a combination of them (especially, Methods 1 and 2). Also, consider these tools below to generate a good quality mesh.

#### **Remove degenerate and duplicate faces**

*Filters → Cleaning and Repairing → Remove Duplicate Faces*

*Filters → Cleaning and Repairing → Remove Duplicate Vertices*

*Filters → Cleaning and Repairing → Remove Zero Area Faces*

...

#### **Remove very small isolated components**

*Filters → Cleaning and Repairing → Remove Isolated Pieces (wrt Face Num.)*

This deletes tiny garbage surfaces.

Set: Min faces = 100 (or higher depending on mesh)

#### **Laplacian Smooth**

*Filters → Smoothing, Fairing and Deformation → Laplacian Smooth*

To improves triangle distribution without distorting shape too much with slight smoothness.

Set these parameters:

Steps: 1–3

Cotangent weighting: ON