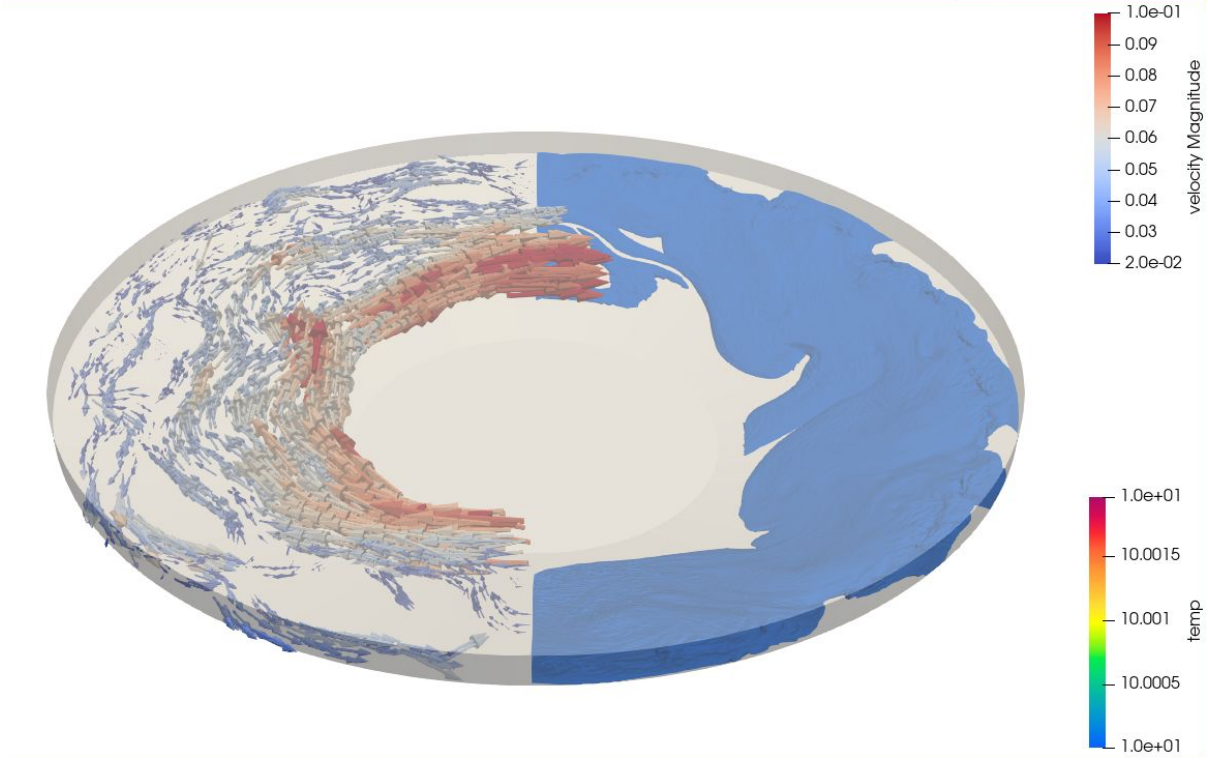
Three-Dimensional visualisation and animation of CROCO numerical data using ParaView

Loïc Merlet, Achim Wirth, Cyrille Bonamy

Three-Dimensional visualisation and animation of CROCO numerical data using ParaView

Manual to visualising 3D results from CROCO simulations, using ParaView software

Loïc Merlet, Achim Wirth & Cyrille Bonamy
loic.merlet@grenoble-inp.org, achim.wirth@univ-grenoble-alpes.fr, cyrille.bonamy@univ-grenoble-alpes.fr
Univ. Grenoble Alpes, CNRS, Grenoble INP, LEGI, 38000 Grenoble, France



SHOM 300 ans d'hydrographie

LEGI

GRENOBLE INP UGA

Table of Contents

List of Figures

1 Introduction

Due to increasing computational capacities smaller and smaller processes can be resolved in numerical models of the ocean dynamics. These processes evolve on scales for which the horizontal and vertical extension become similar. This is also reflected in the demand of non-hydro-static modelling in which the full 3D Navier-Stokes equations rather than the 3D-2Components primitive equations are solved.
The open-source 3D visualization and data analysis interactive application ParaView is an adapted tool to visualize 3D processes in 3 spatial dimensions and time. This document focuses on 3D post-processing of NetCDF files using ParaView software. The NetCDF files are the output files of the code CROCO for ocean numerical simulations.

1.1 NetCDF

CROCO's output files are in the NetCDF format; It is a "self-describing" format, this means that it contains a header which describes the layout of the rest of the file, in particular the data arrays, as well as arbitrary file metadata in the form of name or value attributes. More informations about NetCDF format are provided in Ref. [**?**].

For every snapshot at time t, the data is given in 3 coordinates: x and y in the horizontal directions and s in the vertical direction. In our example, there are 1024 points in the x and y directions, whereas there are 64 points in s. The particularity is that the grid is regular but not cartesian: all its 64 levels follow the topography of the domain. The whole dataset contained in the output file such as temperature, salinity, velocity or acceleration is written in this coordinate system.

1.2 ParaView

Visualization is the process of converting raw data into images and renderings to gain a better cognitive understanding of the data.

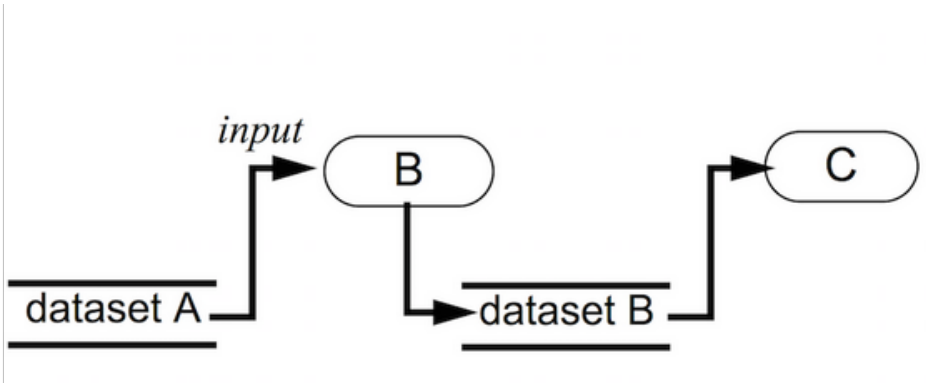


Figure 1: Pipeline of visualization in ParaView

In ParaView, there are data and process objects. Data objects (dataset A and B) represent and provide access to data; process objects (B and C) operate on the data as described in Fig. 1: The dataset A feeds the process object B. B creates a rendering from the dataset A and it outputs a new dataset B. This new dataset B can be processed by a new object C, and so on. Objects B and C can be filters or any tool available in ParaView. Processing can also be performed in parallel pipelines. (See Ref. [**?**] for further information).

ParaView is the graphical front-end to the ParaView application. The GUI (Graphical User Interface) is designed to allow to easily create pipelines for data processing with arbitrary complexity. The GUI provides panels to inspect and modify the pipelines, to change parameters that in turn affect the processing pipelines, to perform various data selection and inspection actions to introspect the data and to generate renderings.

As show in Fig. 2, four main areas can be distinguished in the GUI:

- Area 1: At the top, there are the tools, filters and options to apply to the data.
- Area 2: On the left, there is the Pipeline browser, it gives the tree structure of all the processes and pieces of data used.

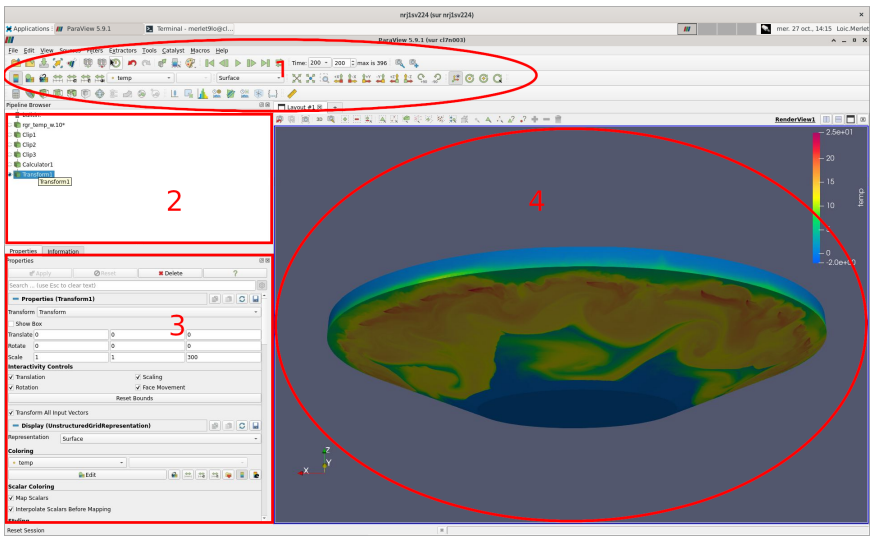


Figure 2: ParaView GUI

- Area 3: Just below, there are the properties and options of the processes to adjust before their application.
- Area 4: Finally, the viewport or central portion of the ParaView window is the area where ParaView renders results generated from the data.

1.3 Adding the vertical coordinate in CROCO's output file

To be able to visualize correctly CROCO output data in 3D there is a first compulsory step: the information of the z-coordinate has to be provided. As the file also contains a variable h(x,y) standing for the depth of the domain at each surface point, and the variables cs(s) and sc(s) that give the vertical stretching of the the grid, the actual depth "gridz" can be calculated for each grid-point by using the following formula:

$$gridz(x,y,s) = \frac{hc * sc(s) + h(x,y)cs(s)}{hc + h(x,y)} h(x,y) \tag{1}$$

As explicated in Ref. [**?**].

Then, the variables available in the CROCO output file (density, temperature, salinity, velocity, acceleration..) have to be extracted and the variable gridz is added to a new netCDF file. As an Arakawa C-grid is used for the variables in CROCO, we either have to use a different grid in x, y, z for the different variables or the variables have to be interpolated on one grid. It can be done using for instance Python or Fortran algorithms.

2 Description of the steps to follow on ParaView to visualize 3D CROCO output data

The steps to follow will be described in this subsection. Constructing the tree structure presented in Fig. 3 provides the 3D visualization. All the filters applied in this tutorial can be found in the "Alphabetical" section, below the "Filter" drop-down menu.

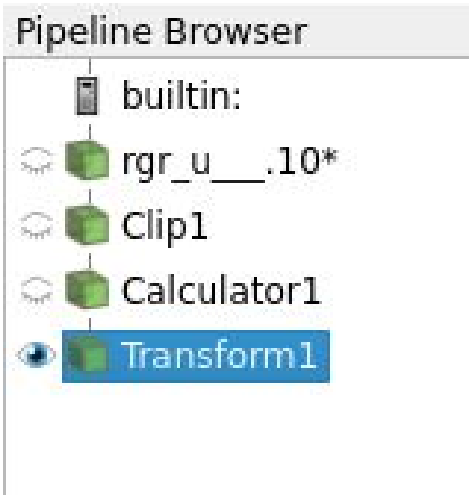


Figure 3: Tree structure of the steps in ParaView

First of all, to load the NetCDF file:

- Use the "File" (Area 1) drop-down menu and "open" your file by clicking on it
- Pick "NetCDF Reader" and click on Ok
- In the properties section (Area 3), make sure to untick the "Spherical coordinates" checkbox
- "Apply" the modifications

Then, as the data are stored in a square domain, whereas the Coriolis platform is a round pool, applying a Clip filter is necessary to remove the corners of the domain so as to only visualize the circular part of the Coriolis platform. To apply it:

- Use the Clip filter (Area 1, line 3, icon 3 or "Filter", "Alphabetical", "Clip")
- Select "Cylinder" as "Clip Type" property (Area 3)
- Place the cylinder "Along the Z axis" without modifying neither its position nor its radius (Area 3)
- "Apply" the Clip filter (Area 3)

It is now possible, thanks to a Calculator filter, to replace the z-axis by the new one "gridz" calculated during the NetCDF modification. To do so:

- Use a Calculator filter (Area 2, line 3, icon 1 or "Filter", "Alphabetical", "Calculator")
- Tick the box "Coordinate results" and use the formula shown in Fig. 4. Remember that the vertical coordinate "gridz" have to be present in the netCDF file
- "Apply" the filter (Area 3)

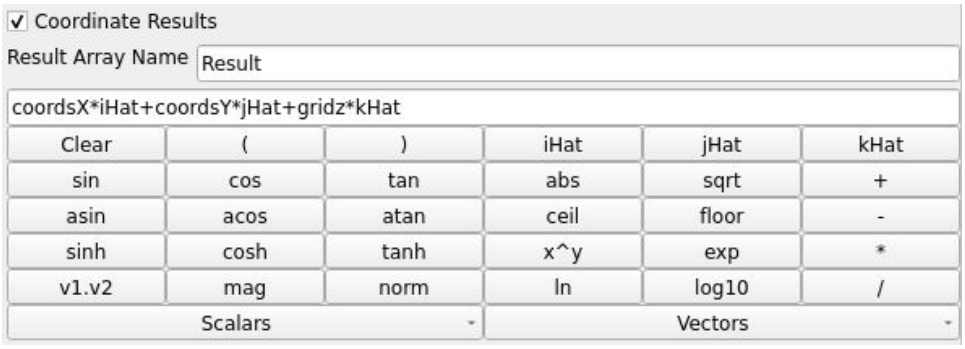


Figure 4: Modifying the z-axis using a Calculator filter

Finally, a Transform filter has to be applied to account for the actual size of the domain. In our case, as the "gridz" coordinate stands for the actual depth of our domain which is 0.51m and the x and y extensions are for now the number of points of the domain, 1024, whereas its actual diameter is 13.5m, a scaling factor in the z direction of approximatively 76 would be appropriate here, however a scaling factor of 300 makes the visualisation more appealing:

- Use a Transform filter (Area 1, "Filters", "Alphabetical", "Transform")
- Use a "scale" factor of 300 in the z direction (last column, "scale" line)
- "Apply" the Transform filter (Area 3)

To display the results and show a scalar at the outer surface of the domain ah shown in Fig. 6, the following settings can be used in the properties panel of the ultimate Transform filter:

- Go down to the "Display panel" (Area 3)
- Select "Surface" as "Representation"
- Under the "Coloring" section, choose the scalar to color the visualisation (for instance the Density anomaly "temp" can be used as shown in Fig. 6)
- It is possible to choose other colors by using the "Choose preset" button

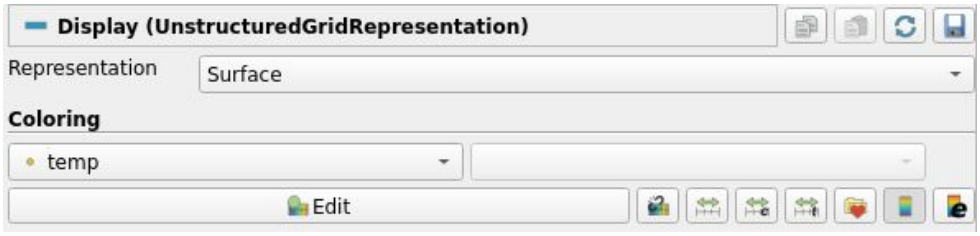


Figure 5: Display menu in ParaView

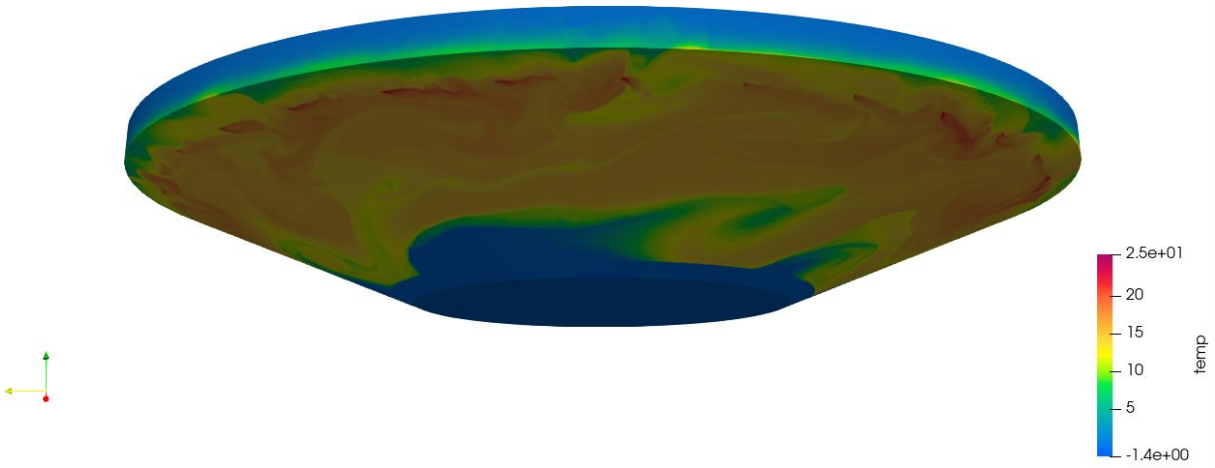


Figure 6: Density anomaly in the Coriolis platform

3 3D Post processing

3.1 Contours of temperature and vertical velocities

The file is now ready to be post processed in ParaView, for instance to visualise a iso-value of the density anomaly as shown in Fig. 7 or both potential temperature and vertical velocities as shown in Fig. 8. To do so:

- Use a contour filter (Area 1, line 3, icon 2 or "Filters", "Alphabetical", "Contour")
- Select the variable to show in the "Contour by" section (Area 3)
- In the "Isosurfaces" section, add the values of the contours to visualise (in Fig. 7, the density anomlay displayed is 10 and in Fig. 8, the velocities displayed are -0.01 and 0.01)
- Click on "Apply"

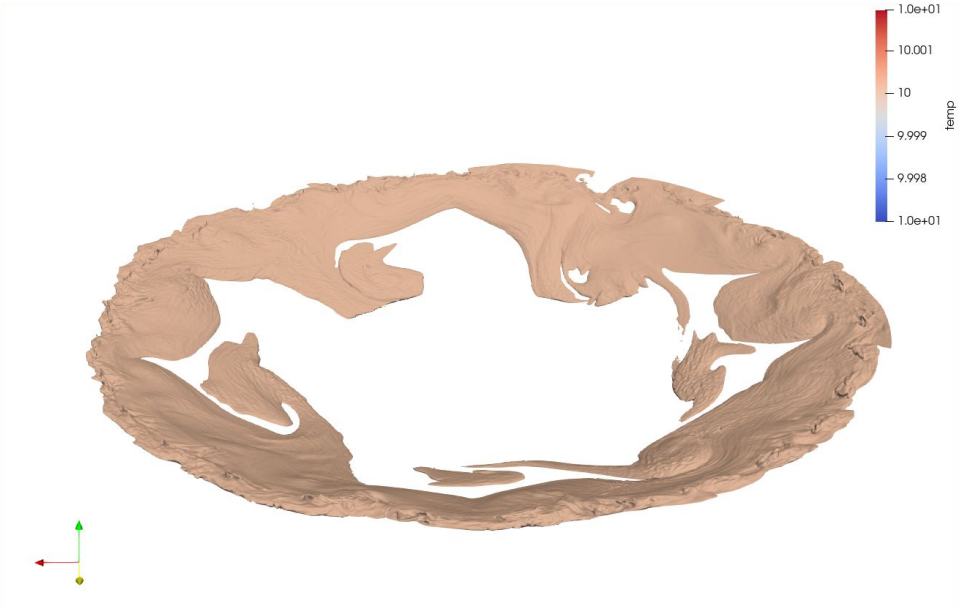


Figure 7: Iso surface of density anomaly

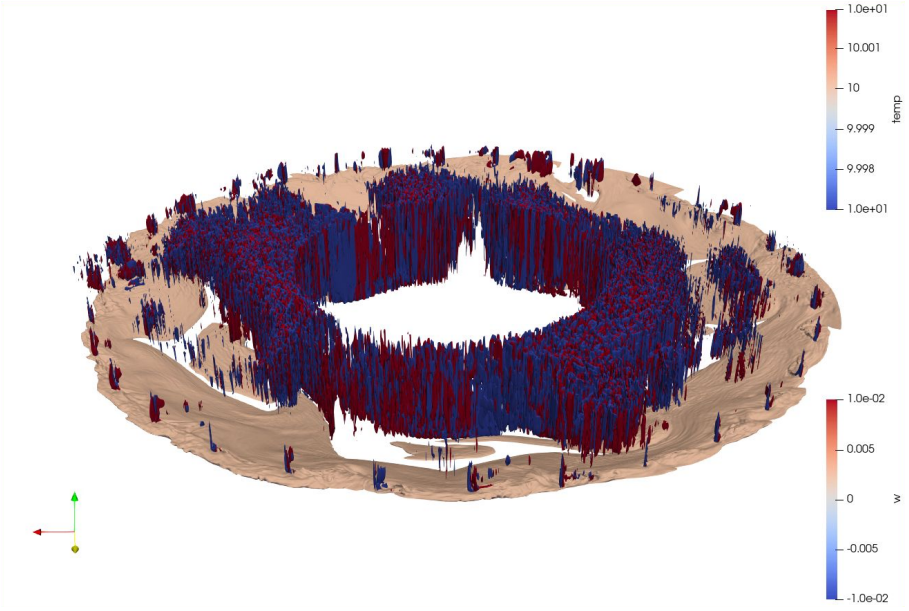


Figure 8: Iso surface of density anomaly and vertical velocities

3.2 Streamlines visualization

Using ParaView, it is also possible to visualize the streamlines as show in Fig. 9, using a file containing the three components of the velocity: u,v and w and the variable "gridz".

Figure 9: Streamlines visualization

Constructing the tree structure presented in Fig. 10 provides the visualization of 3D streamlines.

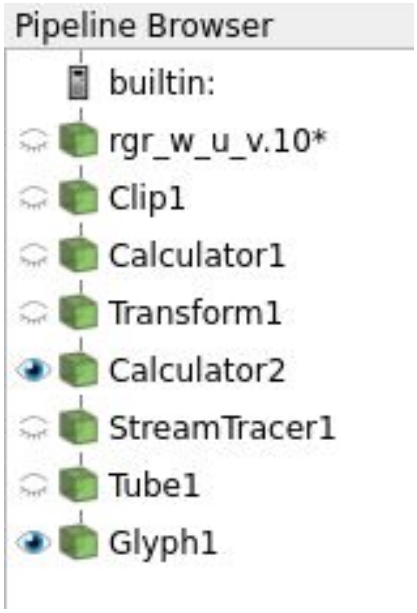


Figure 10: Tree structure for streamlines visualization

When all the steps of section 2 are complete, the first step is to calculate the velocity vector field using a calculator filter:

- Use a Calculator filter (Area 1, line 3, icon 1 or "Filter", "Alphabetical", "Calculator")
- Use the formula in Fig. 11: $iHat * u + jHat * v + kHat * w$ (Area 3) to create the vector field
- Give "velocity vector" as "Result Array Name"
- "Apply" the filter

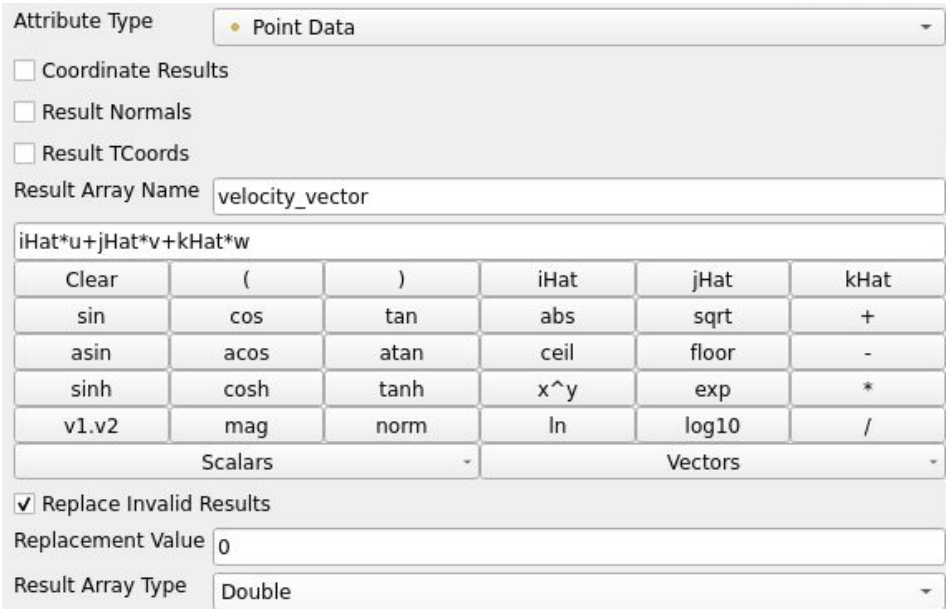


Figure 11: Calculator filter for velocity vector field

Then, a SteamTracer filter has to be applied to plot basic streamlines:

- Use a SteamTracer filter (Area 1, line 3, icon 8 or "Filter", "Alphabetical", "SteamTracer")
- In properties section (Area 3), pick "velocity vector" as "Vectors" to plot
- Pick "line" as "Seed type" in the "Seeds" section

- In the coloring section (Area 3), choose "surface" as representation and to color it by velocity vector and magnitude, respectively in the first and the second drop-down menus to be able to visualize them
- "Apply" the filter

With this first filter, streamlines are introduced, but their appearance can be modified. For instance, a Tube filter allows to adjust their aspect and thickness:

- Apply a Tube filter (Area 1, "Filter", "Alphabetical", "Tube")
- In properties section (Area 3), pick "AngularVelocity" as "Scalars" and "velocity vector" as "Vectors" to plot
- The thickness of the tubes can be adjusted by changing the "Radius" parameter
- In the coloring section, choose "Surface" as representation and color it by "velocity vector" and "magnitude", respectively in the first and the second drop-down menus, to be able to visualize them properly
- "Apply" the filter

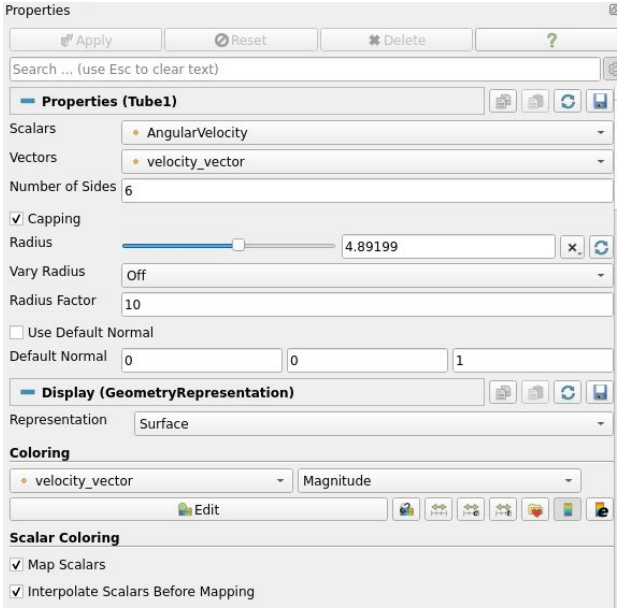


Figure 12: Tube filter parameterization

Finally, with a Glyph filter it is possible to visualize arrows as shown in Fig. 9:

- Apply a Glyph filter (Area 1, line 3, icon 7 or "Filter", "Alphabetical", "Glyph")
- Choose "Arrows" as "Glyph Type" (Area 3)
- Pick "Velocity Vector" as "Orientation array" and "Scale Array" as shown in Fig. 13
- It is possible to adjust the size of the arrow using the "Scale" factor
- In the coloring section, choose "Surface" as representation and color it by "velocity vector" and "Magnitude", respectively in the first and the second drop-down menus to be able to visualize them properly
- "Apply" the filter

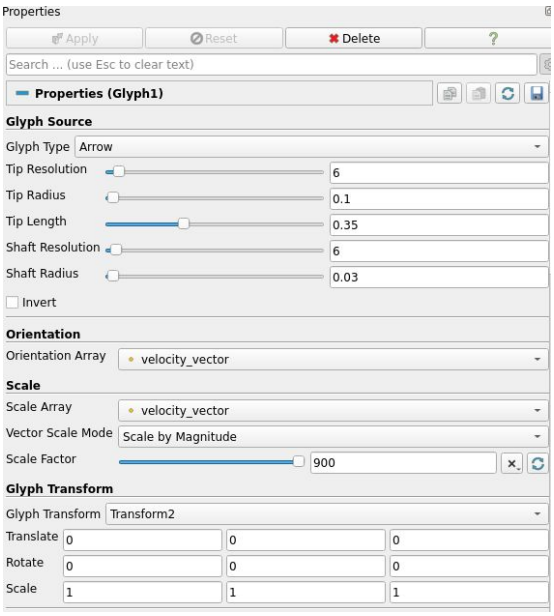


Figure 13: Glyph filter parameterization

3.3 Parallel pipeline: visualization of both contours and streamlines in half of the domain

It is also possible to visualize several variables or filters at the same time, as show in the Fig. 14, dividing the domain and visualizing a different thing in each partition thus created. To do so, after the last step of the second part of this document (the transform filter), for instance to separate your domain in two parts:

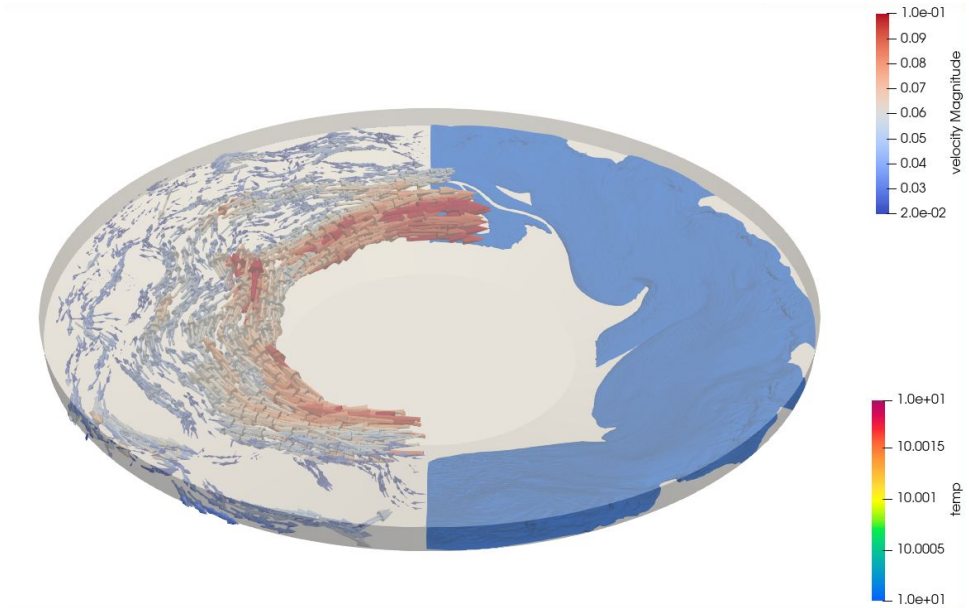


Figure 14: Both contour and streamlines visualization

- Apply a Clip filter (Area 1, line 3, icon 3)
- Choose box as "Clip type" (Area 3)
- Reduce the x or y extension of the box by half as shown in Fig. 15
- "Apply" the Clip filter
- Repeat these steps with the other half of the domain

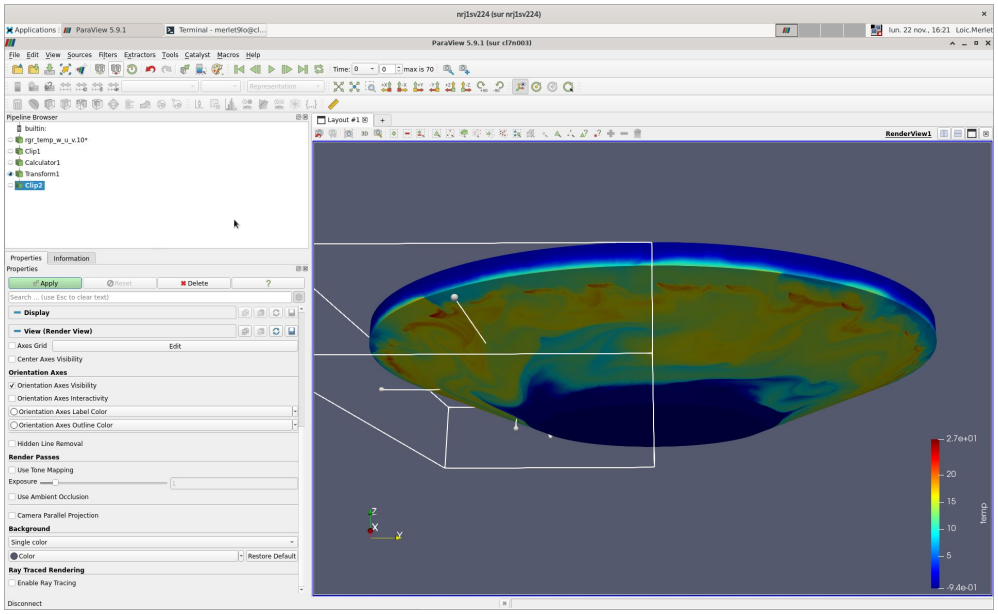


Figure 15: Applying the first clip filter to separate the domain in two parts

It is now possible to apply the method of parts 3.1 and 3.2 selecting one by one the two Clip thus created into the tree structure to obtain a parallel visualization as shown in Fig. 16 to both visualize contours of temperature and streamlines for instance, as shown in Fig. 14.

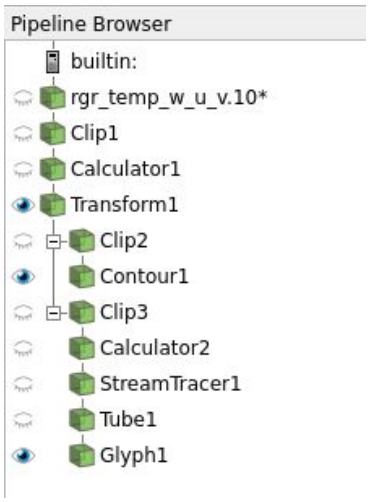


Figure 16: Parallelised tree structure in ParaView

3.4 Scripting function in ParaView

With ParaView, an interesting function is the scripting. Indeed, the "state" of the visualization, which includes acces to the files, all the filters applied to it and all the details of the visualization, can be described in a script. This script can then be loaded at a later application for reuse or modification. The easiest way to use this functionality is to save the state in the ".pvsm" format and to load it back afterwards:

- Use the "Save state" option under the "File" drop-down menu (Area 1)
- Choose the ".pvsm" or the ".py" format
- The state is stored and can then be loaded using the "Load state" option in another ParaView instance, using the "File" drop-down menu.

Saving the state in ".pvsm" or Python ".py" format allows to modify the script, for instance to apply the same sequence to another file or series of files. It can also be used to automatically process data and generate renderings.

4 Film making: creating animations

To animate the visualisation it is interesting to appreciate qualitatively the temporal evolution of the phenomena studied. ParaView default animation keeps the filters on and pushes the time forward. It can be enhanced with different options such as camera movements.

- Click on the drop-down "View" panel (Area 1)
- Activate the "Animation view" section
- A menu appears as shown in Fig. 17, in which parallel options can be added

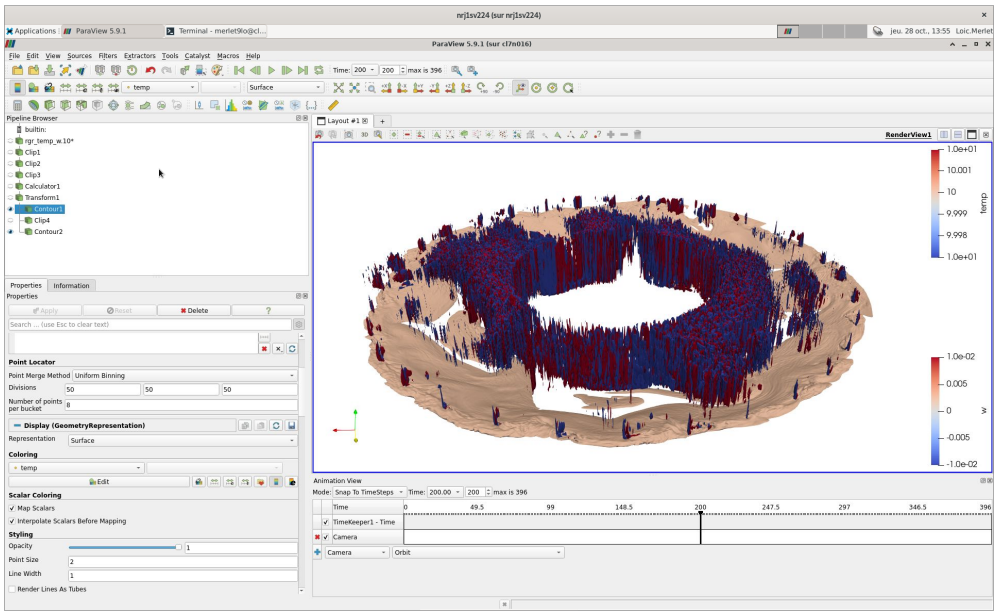


Figure 17: Animation view panel

When the animation is ready, the following steps allow to save the animation:

- Choose "Save animation" in the "File" drop-down menu (Area 1)
- The options of the animation such as its resolution, its format, the frame rate or the frame window can now be modified as shown in Fig. 18

Acknowledgements: The work was funded by Shom, contract N 20CP03.

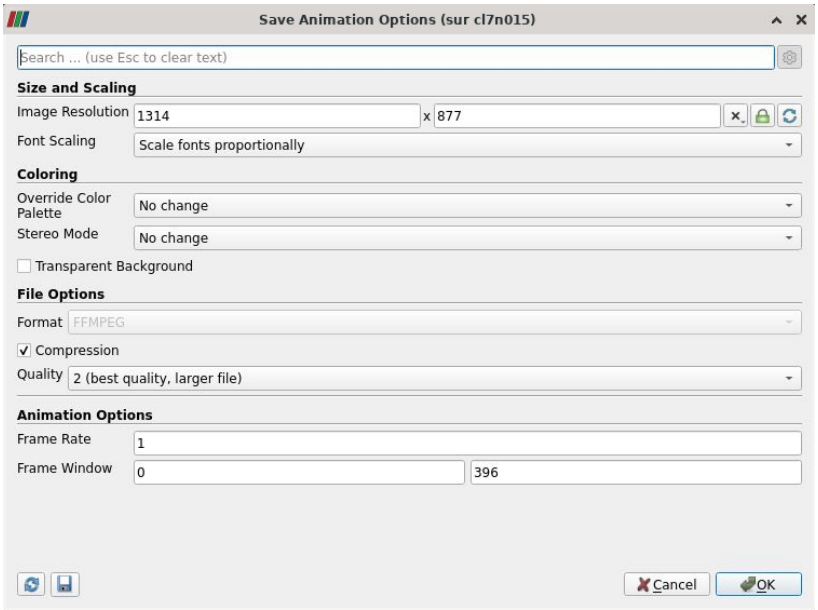


Figure 18: Save animation options