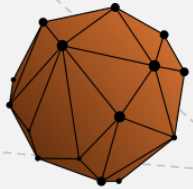
FOAM@Iberia 2023 5th edition

Post Processing in ParaView

An overview

Gabriel Mar Pinto Wagner
gmpwagner@gmail.com
p42786@alunos.uminho.pt



Computational
Rheology
@IPC

University of Minho
School of Engineering

iPC
INSTITUTE FOR
POLYMERS AND COMPOSITES

Contents

1

Introduction

- What is ParaView?
- History
- Basics of visualization
- Why is it important for OpenFoam users?

What is ParaView?

- An open-source application for visualizing two and three dimensional data sets.
  - It is designed to handle complex scientific and engineering data.
    - Offers a flexible and intuitive graphical user interface.
- Widely used by many academic, government and commercial institutions.
  - Downloaded roughly 100.000 times every year.
    - Developed and maintained by Kitware Inc.

History

2000 - 2002

2000 – ParaView project started as a collaborative effort between Kitware Inc. and Los Alamos National Laboratory.

2002 – The first public release, ParaView 0.6, was announced in October 2002.

2005 - 2007

2005 – In September ParaView 3.0 started being developed focused on a more user-friendly interface.

2007 – ParaView 3.0 was released in May 2007.

2013 - 2016

2013 – ParaView 4.0 was released in June 2013, which introduced more cohesivie GUI controls and better multiblock interaction.

2016 – ParaView 5.0 was released in January 2016, which provided a major update to the rendering system.

2016 > now

From that on, ParaView development continued until today, focused on implementing new features, graphical optimizations.

2023 – Latest release: ParaView 5.11.2

Basics of visualization

3 basic steps to visualize your data

Reading > Filtering > Rendering

Firstly, data must be read in Paraview.

Next, you may apply any number of filters that process the data to generate extract or derive features from the data.

Finally, a viewable image is rendered from the data.

Why is it important for OpenFOAM users?

Data Visualization: Allows the visualization of complex simulations in OpenFOAM, making it easier to understand and explore data results.

Advanced Analisys: It offers a wide range of filters and tools to perform in-depth analysis, extract insights and validade simulations.

Customization: Scripting capabilities allow users to create custom workflows, automate tasks and integrate with OpenFOAM simulations.

Efficiency: It accelerates the post-processing workflow, saving time and resources.

2

User interface

- Overview
- Menu bar
- Toolbars
- Pipeline browser
- Properties panel
- Canvas

Overview

Menu bar

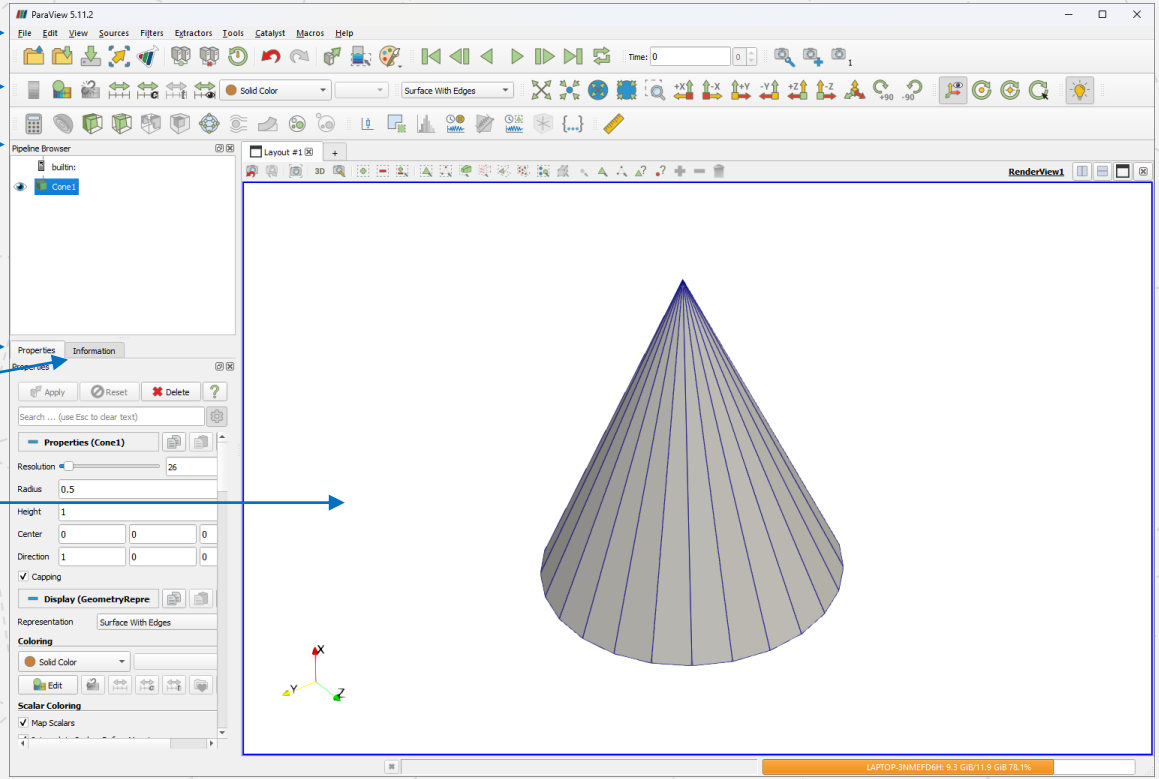
Toolbars

Pipeline Browser

Properties panel

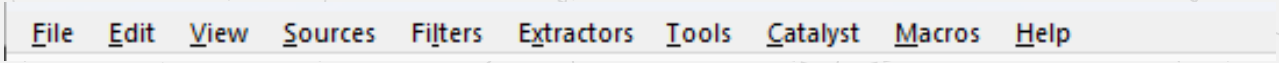
Information panel

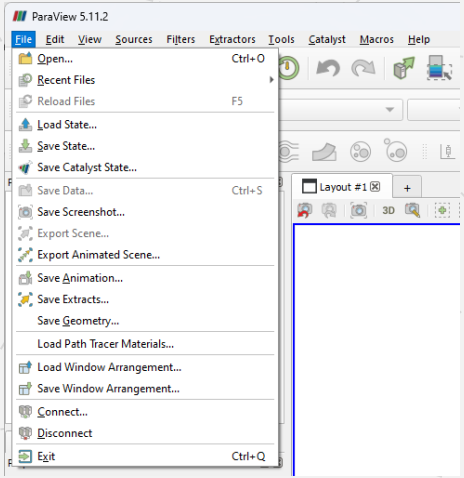
Canvas

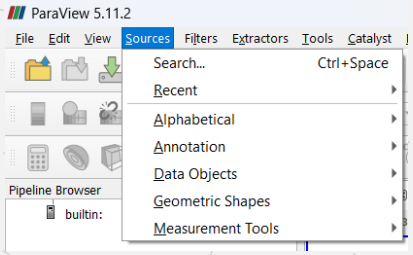


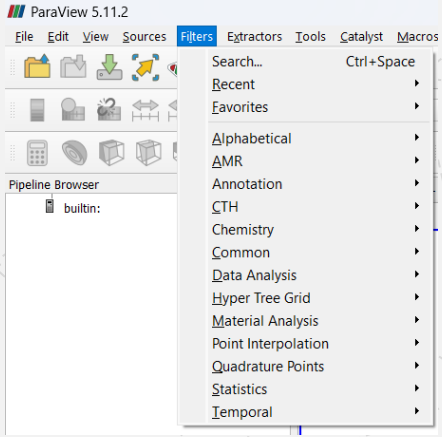
Menu bar

The menu bar allows you to access the majority of tools and features, dividing it into sections.



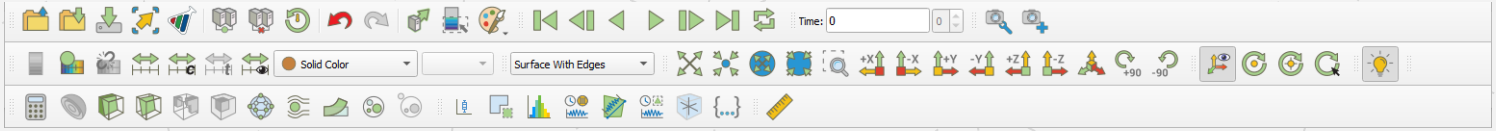






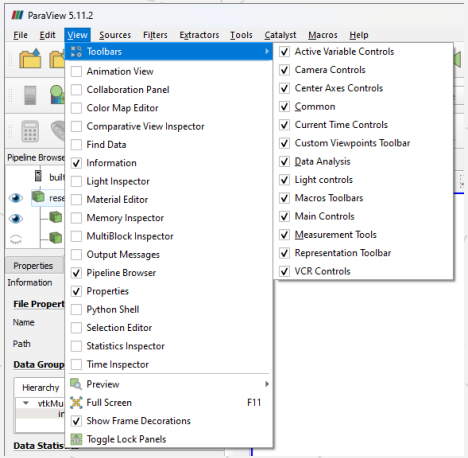
Toolbars

The toolbars provide shortcuts to some features and tools.



Toolsbars are customized by hiding/showing what you desire.

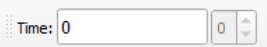
Menu bar *View* - *Toolbars*



Toolbars

Main Controls

VCR Controls



Current Time Controls

Custom Viewpoints Toolbar

Representation Toolbar

Active Variable Controls

Camera controls

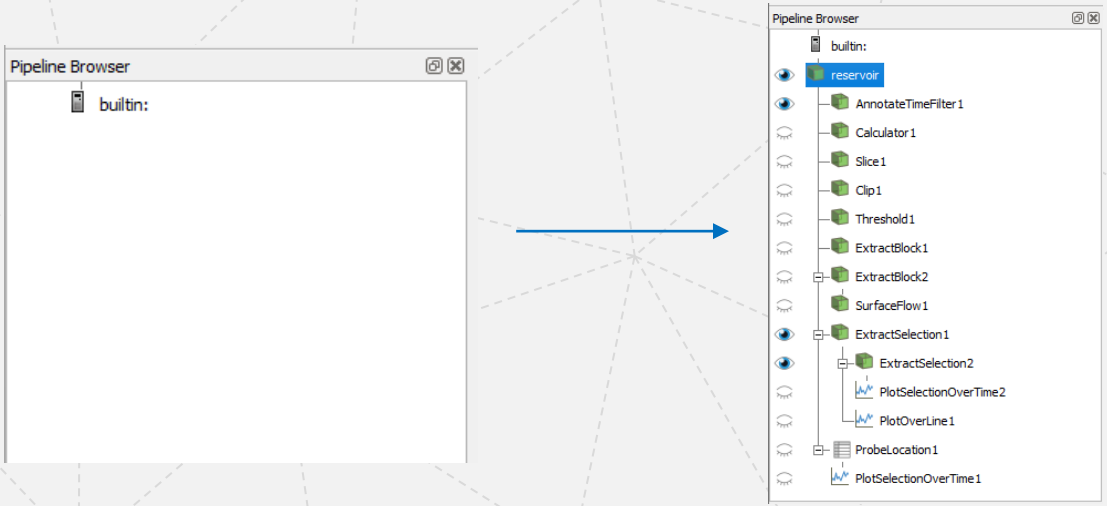
Center Axes Controls

Common Filters

Data Analysis Toolbar

Pipeline browser

The pipeline browser shows the pipeline structure, providing the objects being post processed and its correspondent applyed filters.

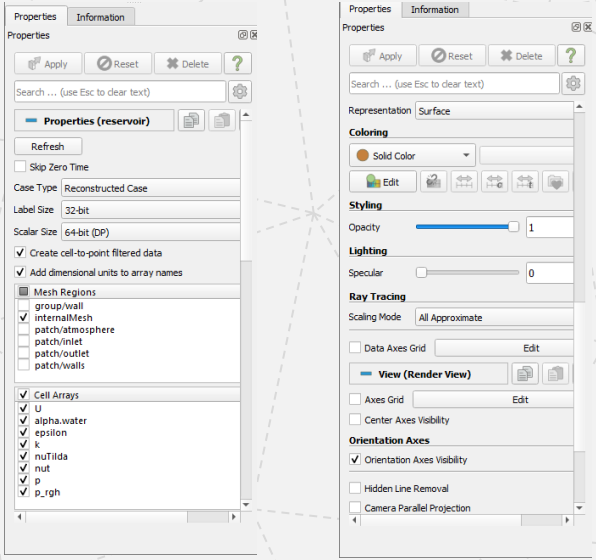


The  icon refers to the objects/filters being shown in the canvas

The  icon refers to the objects/filters being hidden in the canvas

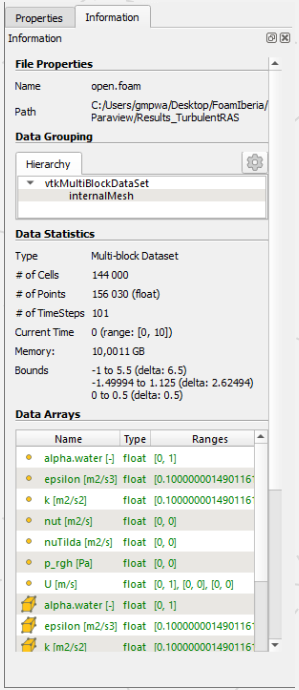
Properties panel

The properties panel allows you to view and modify some parameters of the selected pipeline object/filter.



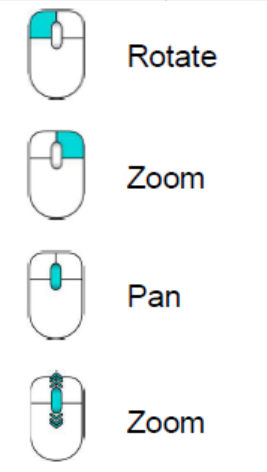
Information panel

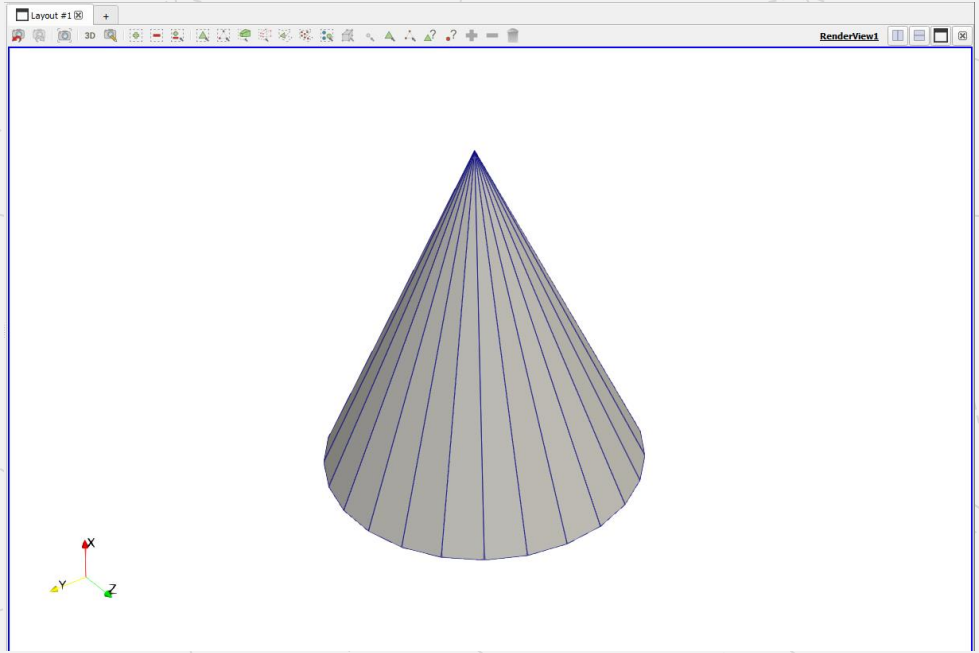
The information panel allows you to visualize parameters information of the selected pipeline object/filter.



Canvas

The canvas allows you to visualize results.





3

Basic Usage

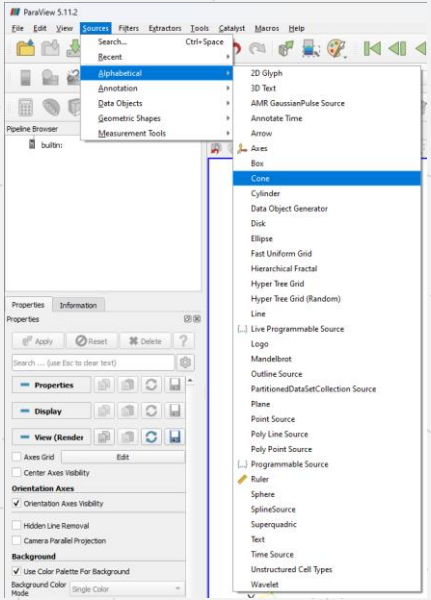
- Creating sources
- Applying filters

Creating sources

Sources refers to specific modules or objects that generate or provide data for visualization and analysis.

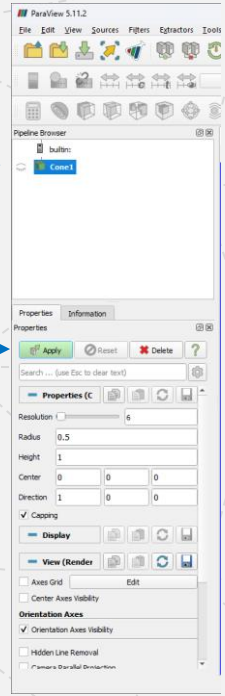
Every available source can be seen in the menu bar *Sources* > *Alphabetical*

They are very handy to explore Paraview's features.



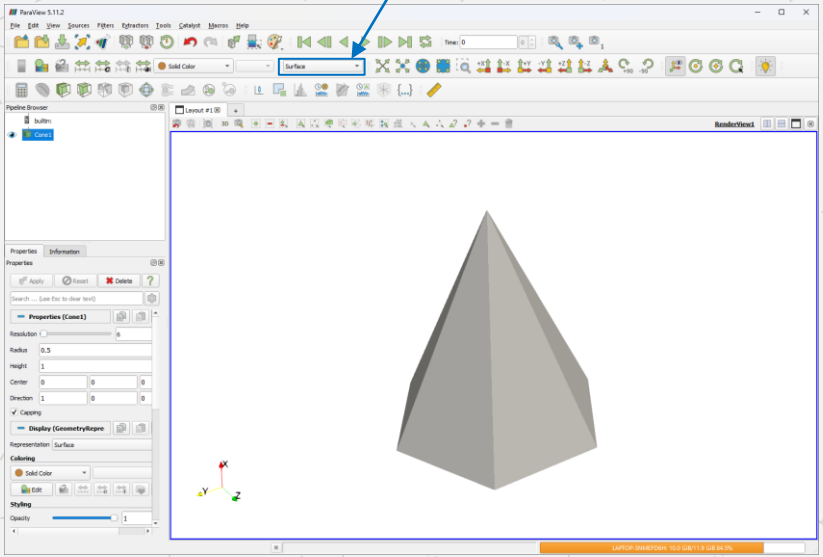
Object "Cone1" will be created

Click Apply

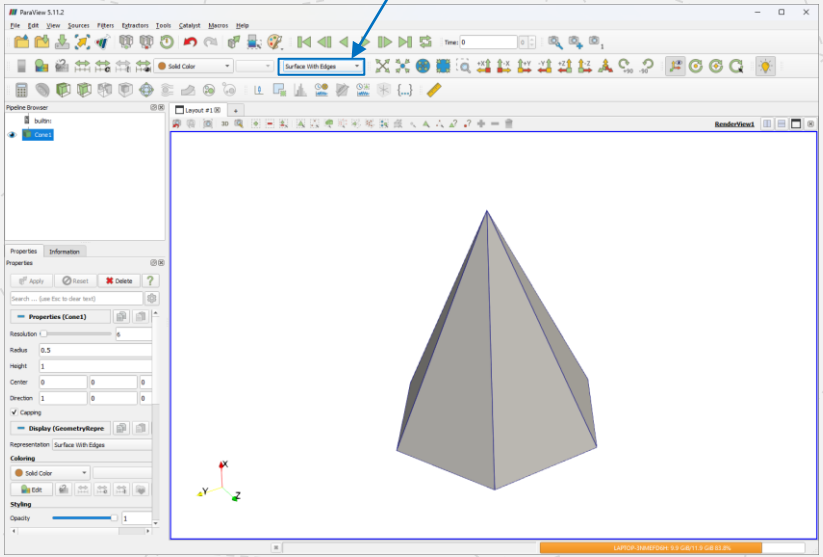


Creating sources

Surface representation

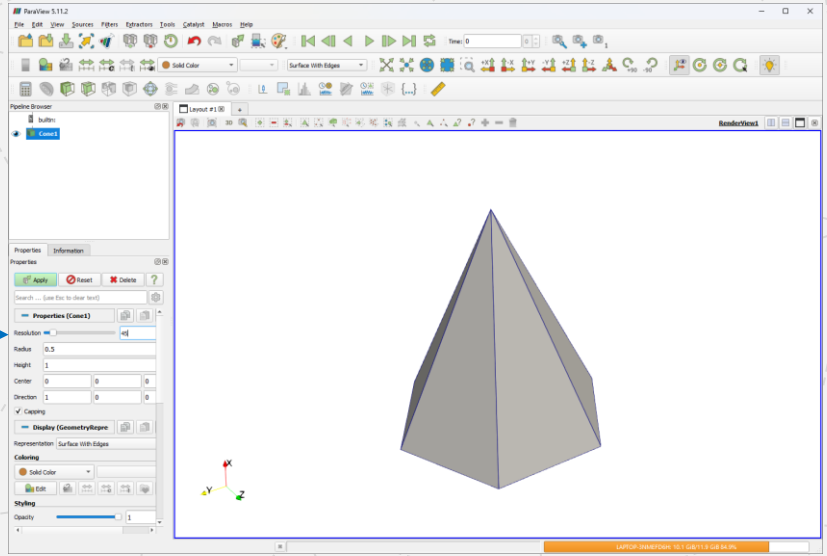


Surface with Edges representation

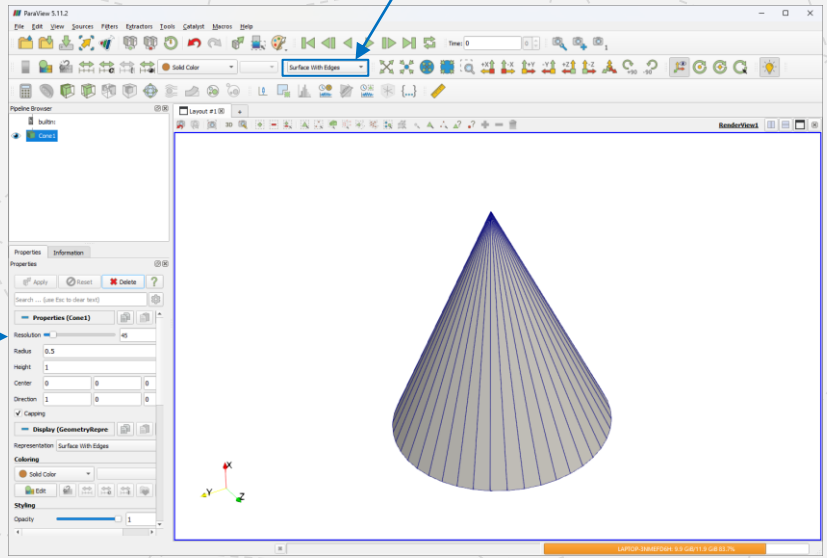


Creating sources

Change “Resolution” to 45 and click 



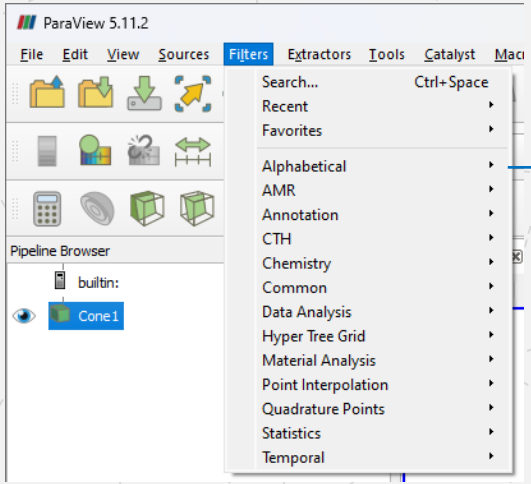
Surface with Edges representation

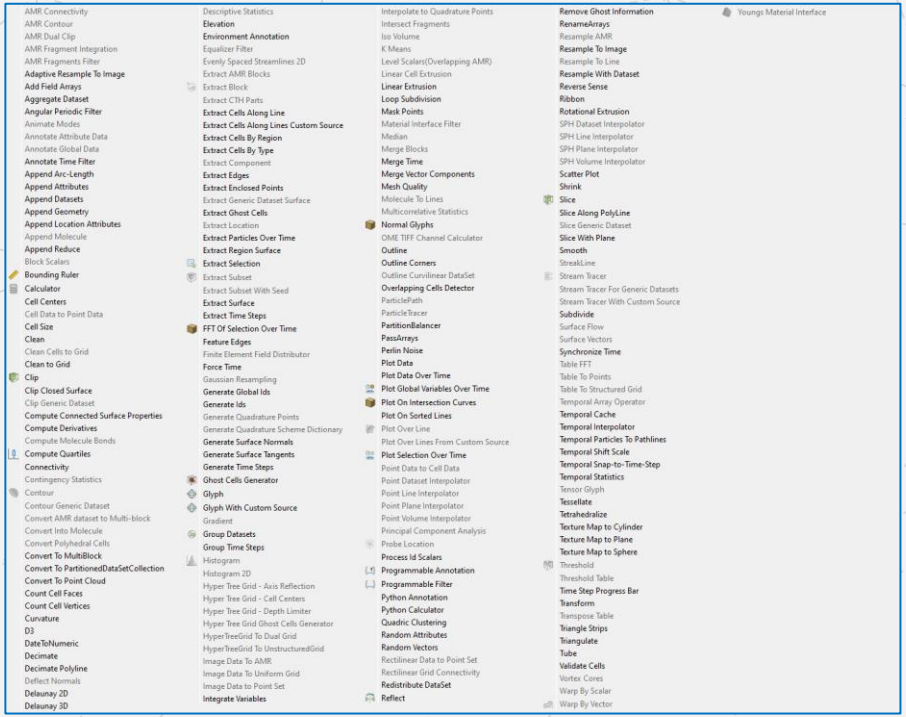


Applying filters

Filters are a set of operations or processing tools that allow you to manipulate, analyse and visualize your data.

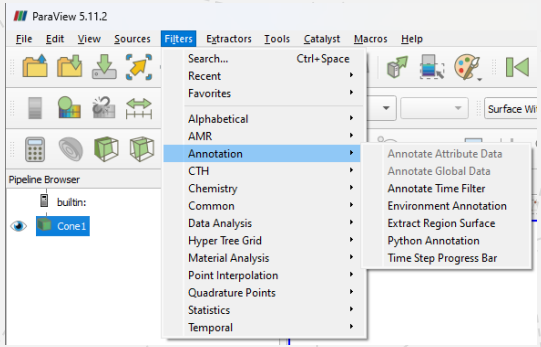
Every available filter can be seen in the menu bar *Filters* > *Alphabetical*

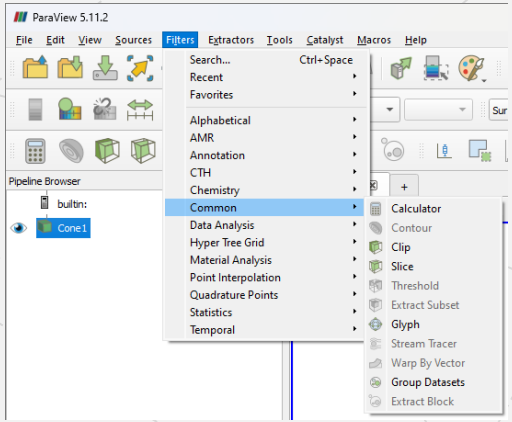


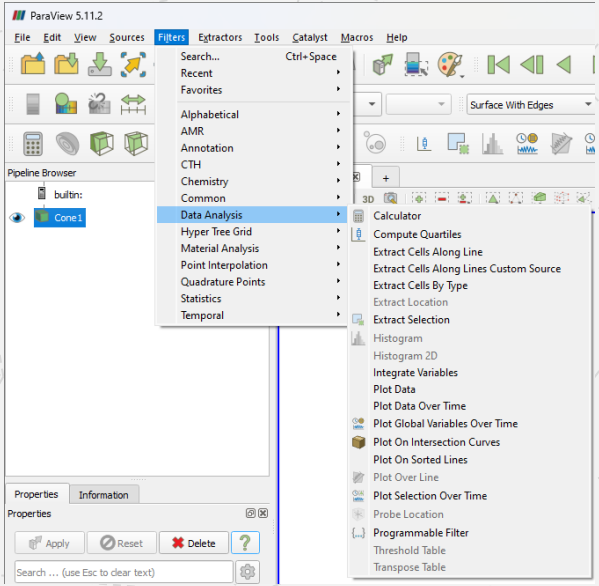


Applying filters

Likewise for the sources, in the menu bar *Filters* every available filter in the *Alphabetical* section is also presented in its type section.



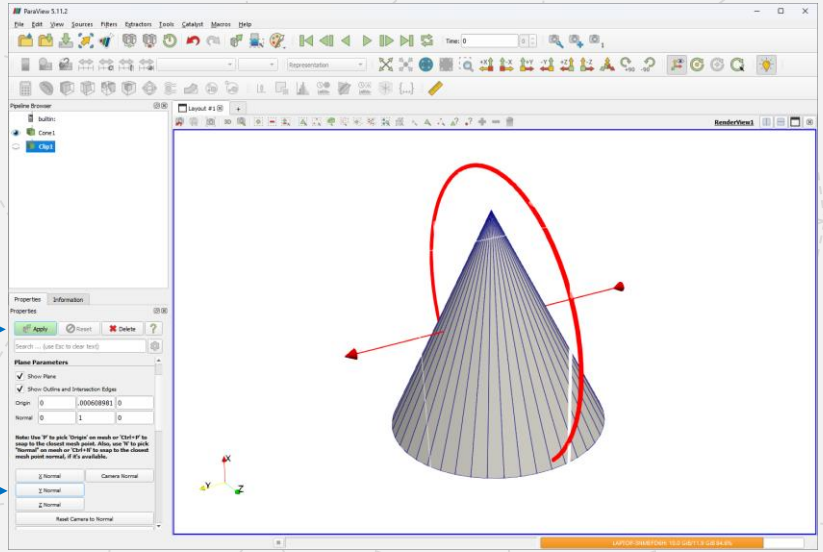


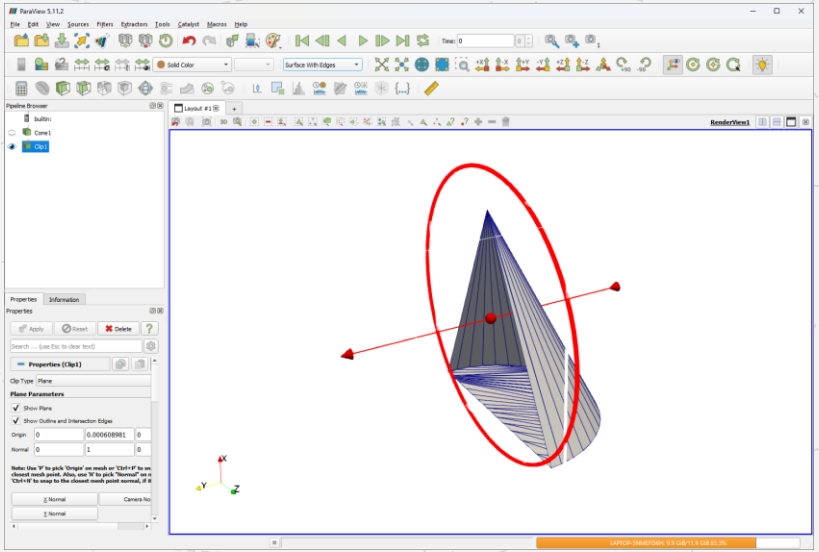


Applying filters

Select the “Clip” filter from the menu bar *Filters* > *Common* > *Clip*

Click 





Define Y normal

4

Importing data

- Loading OpenFOAM simulation data
    - From terminal
    - Directly from Paraview
- Mesh visualization

Loading OpenFOAM simulation data

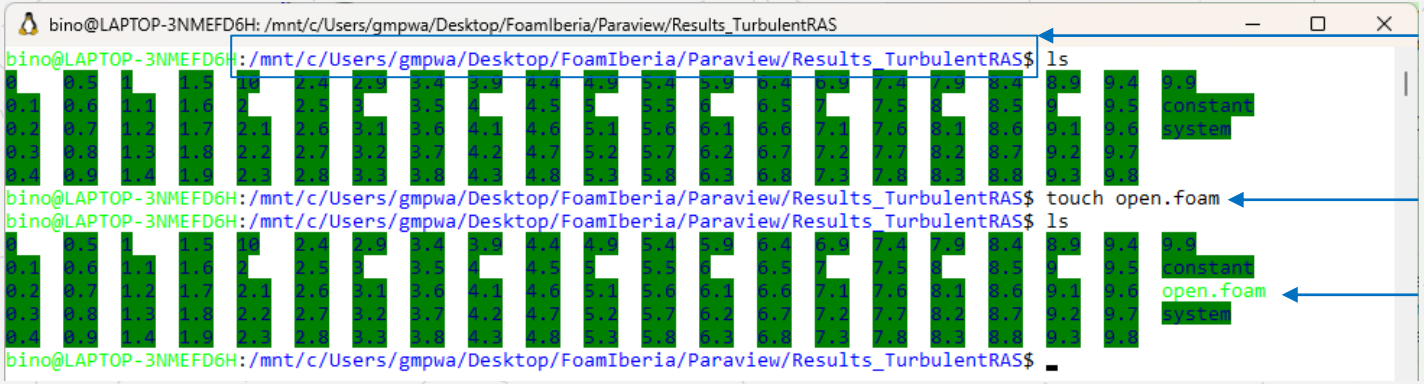
Paraview reads an OpenFOAM case results through a ".foam" file extension.

This file can be created in the terminal by the following command:

$CASE > touch open.foam

Inside an OpenFOAM case folder

Can be replaced for anything you like since you keep ".foam" extension (e.g. blabla.foam)

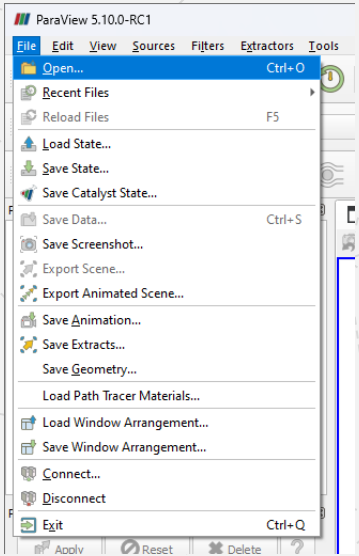


Loading OpenFOAM simulation data

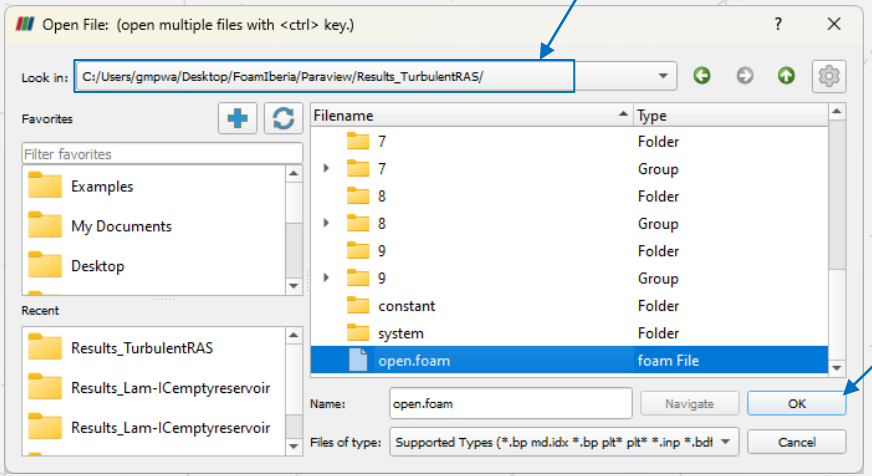
From Paraview

In the toolbar: File > Open…

- Select the ".foam" file extension inside an OpenFOAM case



OpenFOAM case folder



Click "Ok"

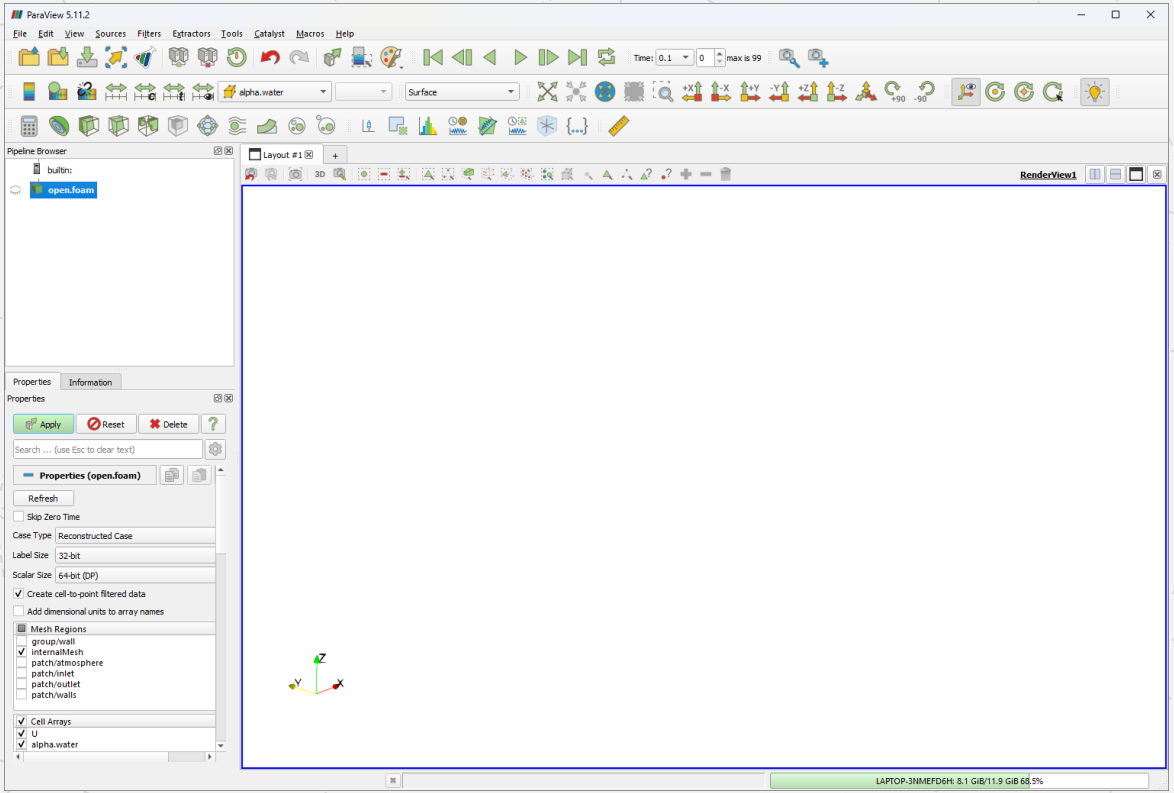
Loading OpenFOAM simulation data

From Paraview

"open.foam" is the created object in the pipeline browser

Click Apply to load data

Uncheck "Skip Zero Time" to enable inital time step visualization



Loading OpenFOAM simulation data

From terminal

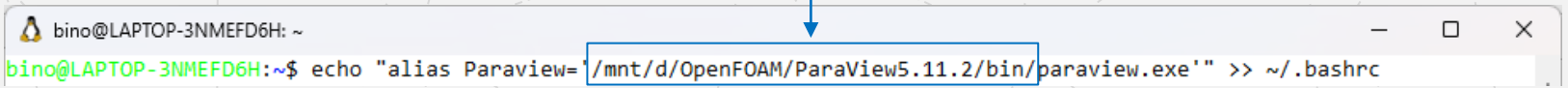
Create an alias called “Paraview” to execute Paraview application.

Use the following command from your terminal:

> echo “alias Paraview=‘$PATHtoAPPLICATION/paraview.exe’” >> ~/.bashrc

Folder where paraview.exe is located

e.g. Folder where Paraview application is located in my computer



Line command

Load .bashrc modifications with “> source ~/.bashrc” command:

```
bino@LAPTOP-3NMEFD6H:~$ source ~/.bashrc
```

← Line command

Loading OpenFOAM simulation data

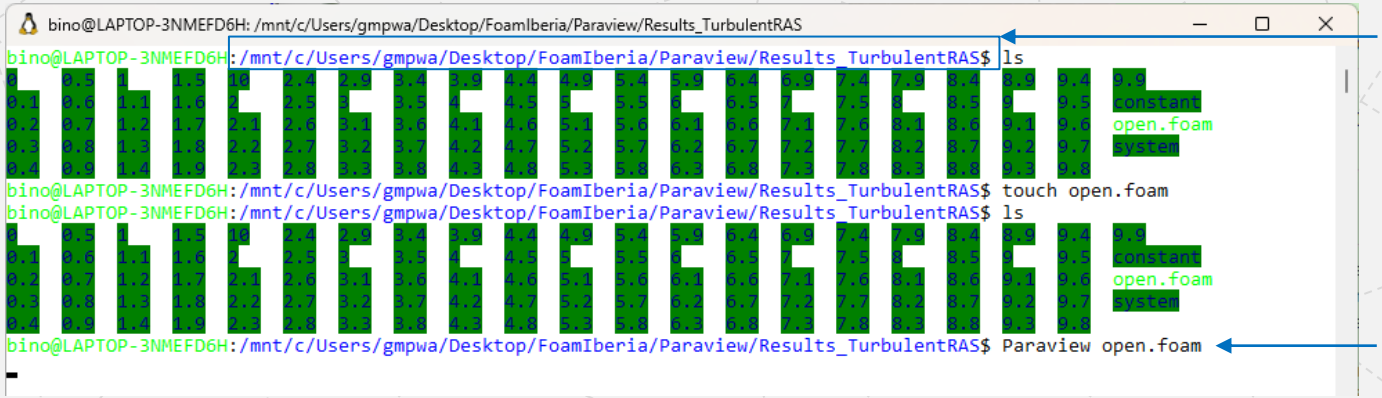
From terminal

Use the created alias pointing to Paraview executable file, followed by the ".foam" file you want to open.

This should be done by the following command:

$CASE > Paraview open.foam

Inside the OpenFOAM case folder

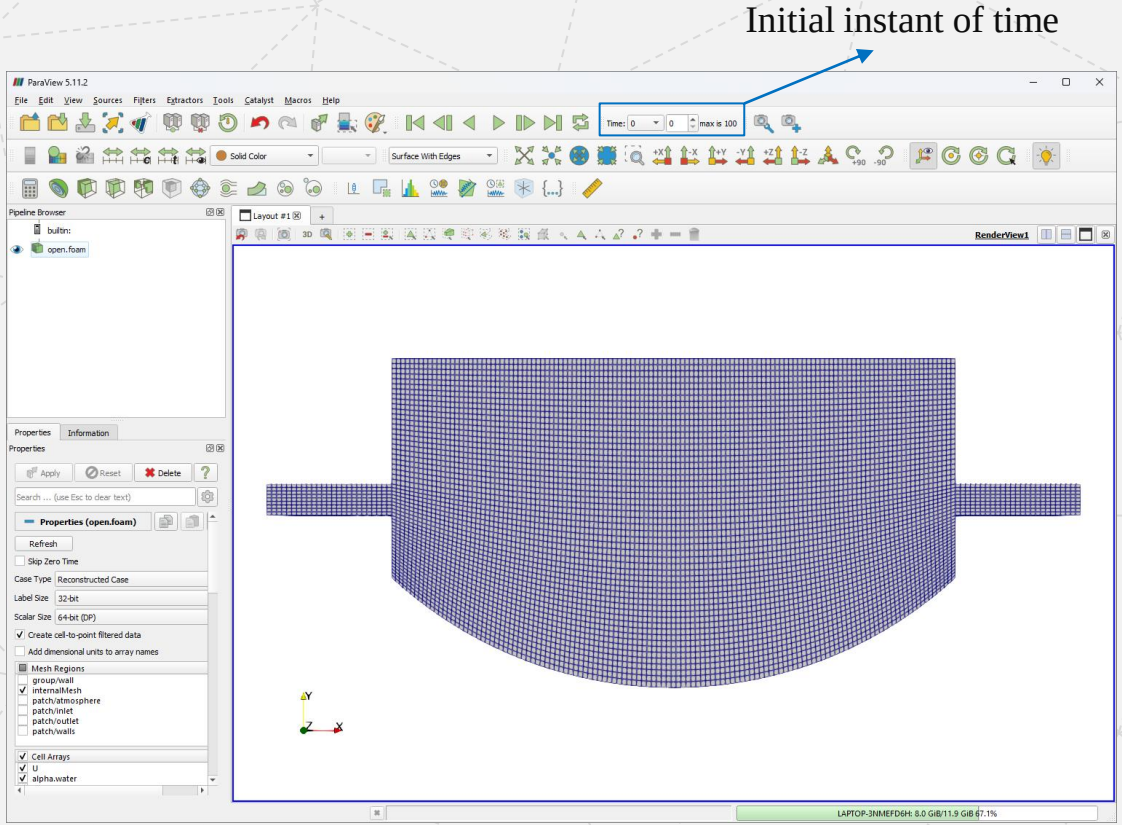


Case folder

Line command

(Paraview open.foam)

Loading OpenFOAM simulation data

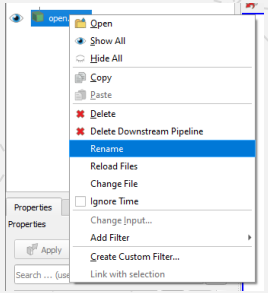


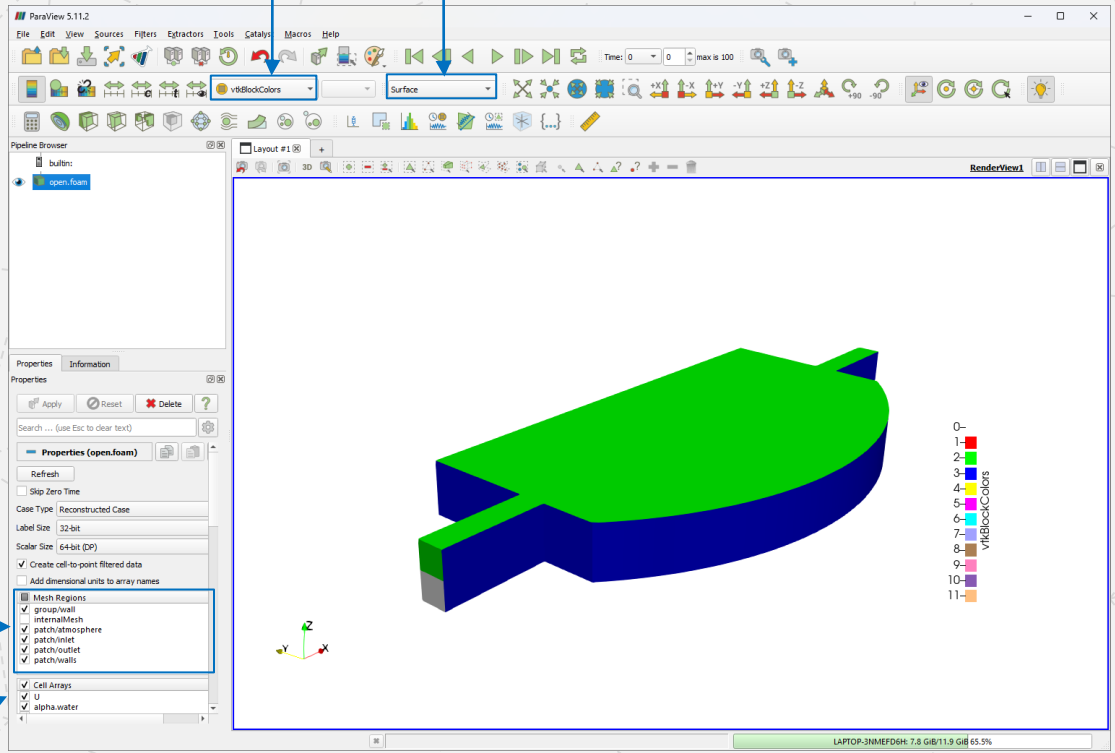
Loading OpenFOAM simulation data

Select vtkBlockColors to visualize patches in different colors

Select surface

Right click on the object and rename it to “reservoir”

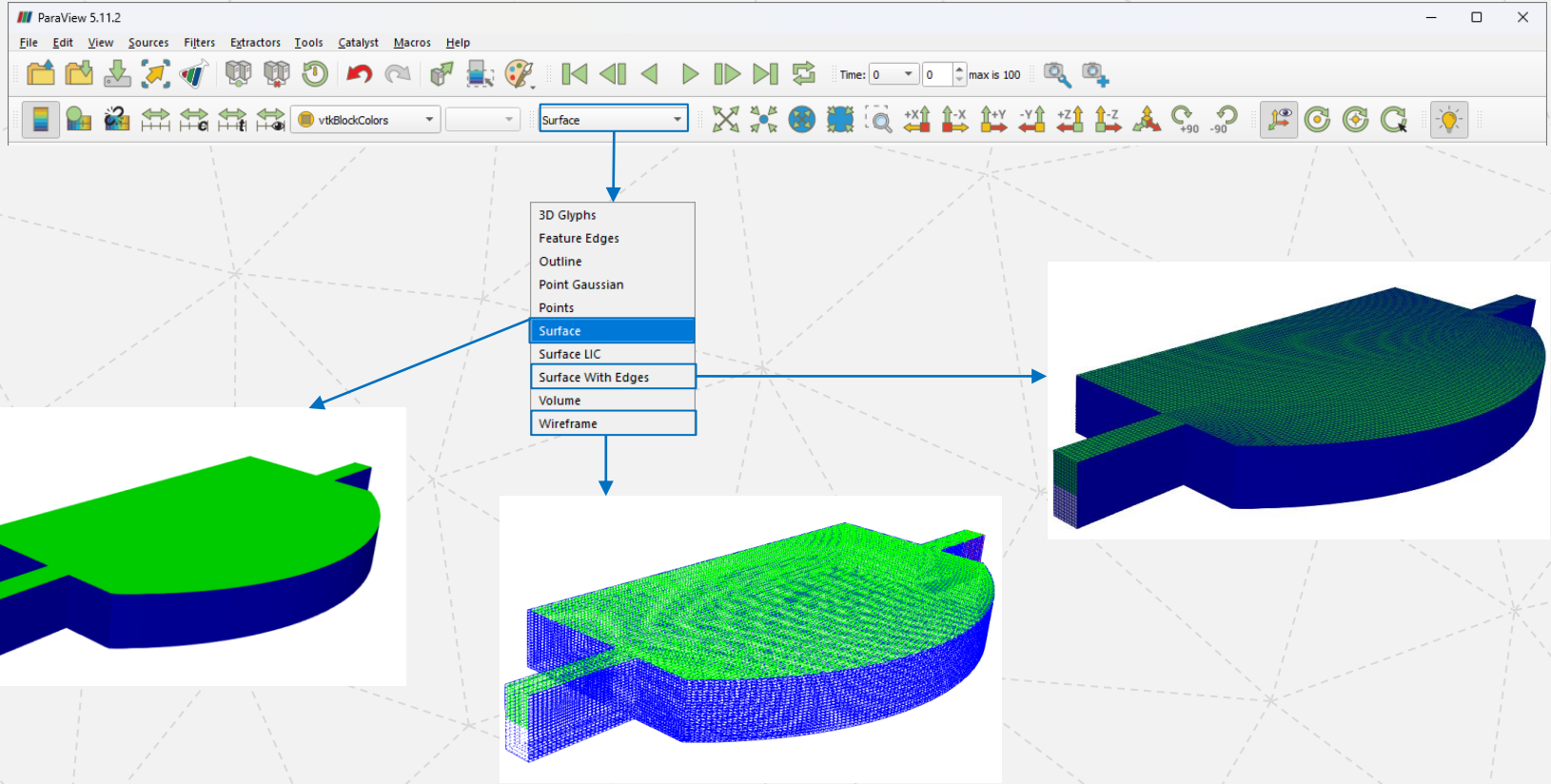




Uncheck internal mesh and check all other options

Check all Cell Arrays

Loading OpenFOAM simulation data

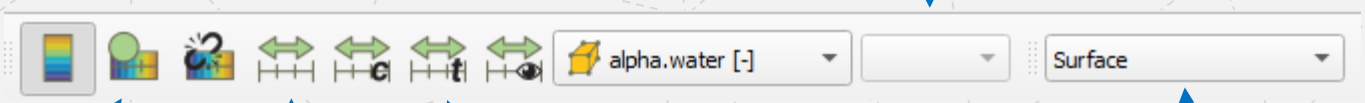


5

Post processing

- Displaying scalar and vector fields
- Adjusting colors, data range and labels
- Applying Common Filters
- Select cells and show values
- Plot over line
- Probe location
- Plot selection over time

Adjusting colors, data range and labels



Toggle color legend visibility

Use separate color map

Rescale to custom data range

Rescale to visible data range

Vector component

Just available when selected field is a vector

Color map editor

Rescale to data range

Rescale to data range over all timesteps

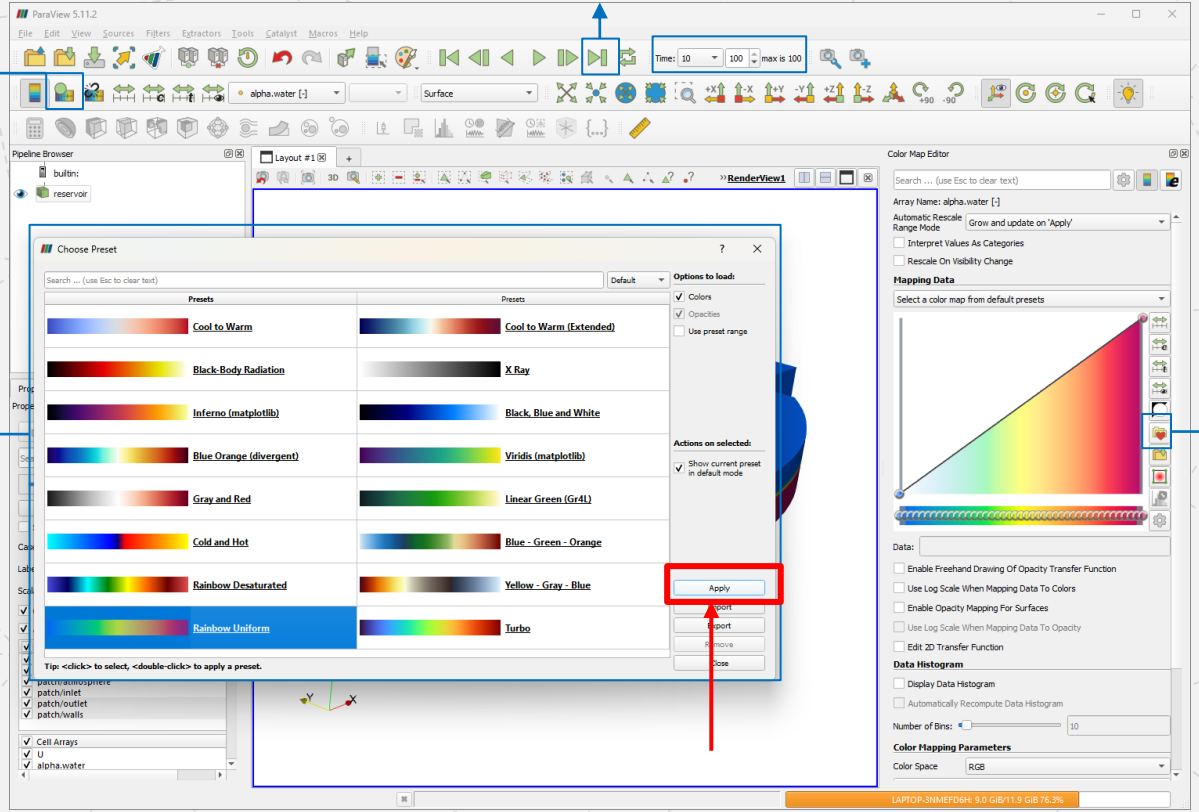
Field being visualized

Object representation

Adjusting colors, data range and labels

Move to last timeStep

1 – click on Edit color map

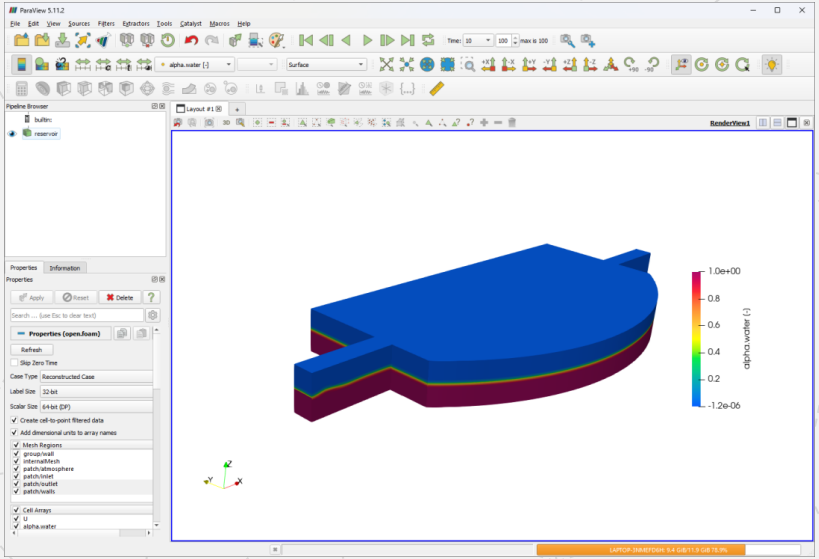


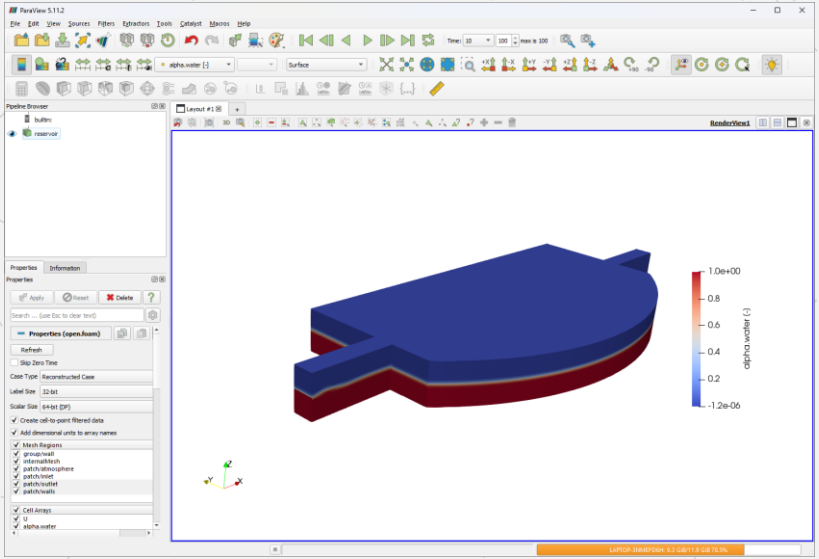
3 – select the color map and click Apply

2 – click on Choose Preset

Adjusting colors, data range and labels

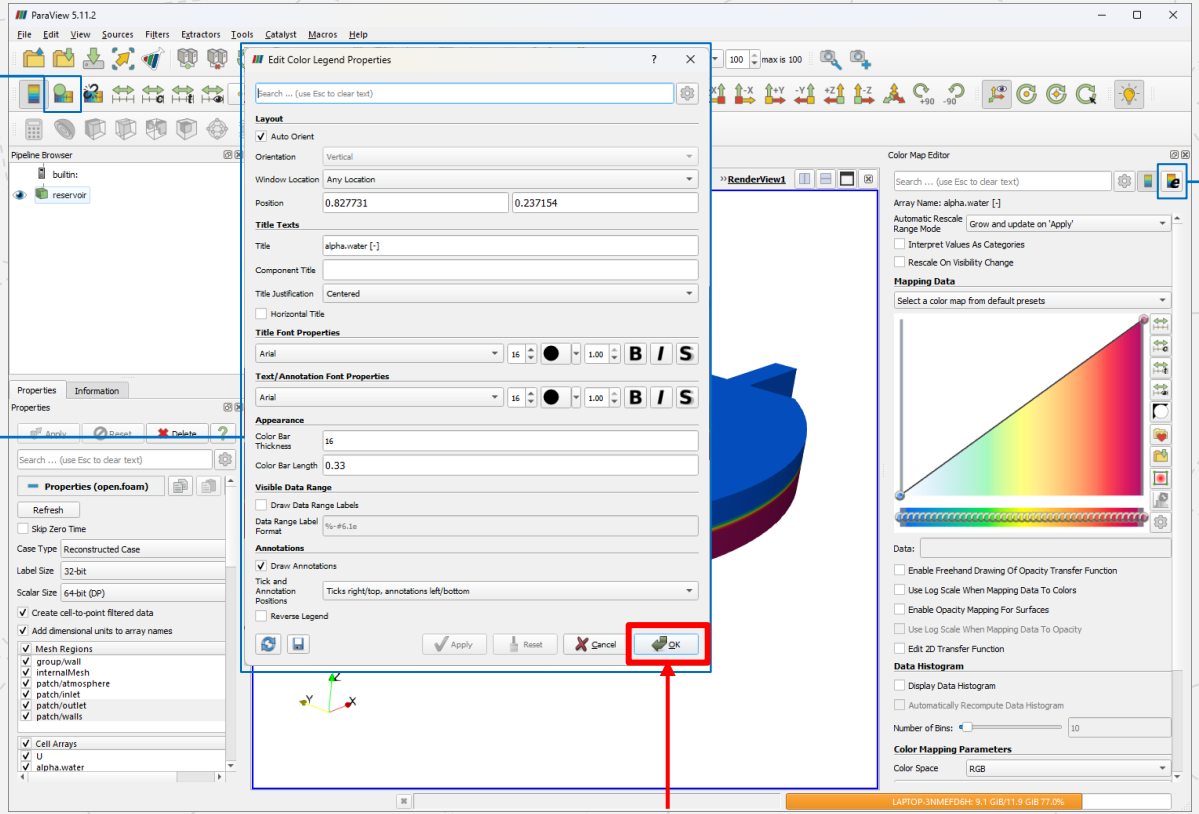
Using different color maps





Adjusting colors, data range and labels

1 – click on Edit color map

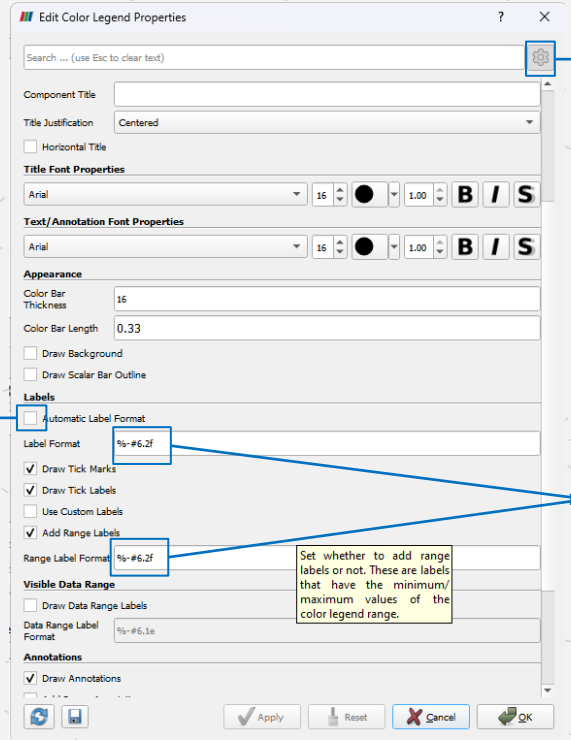


2 – click on Edit color legend properties

3 – define color legend properties and click Ok

Adjusting colors, data range and labels

Option to edit Data Range label format



1 – Go to the gear

2 – Uncheck it

3 – Define %-#6.2f

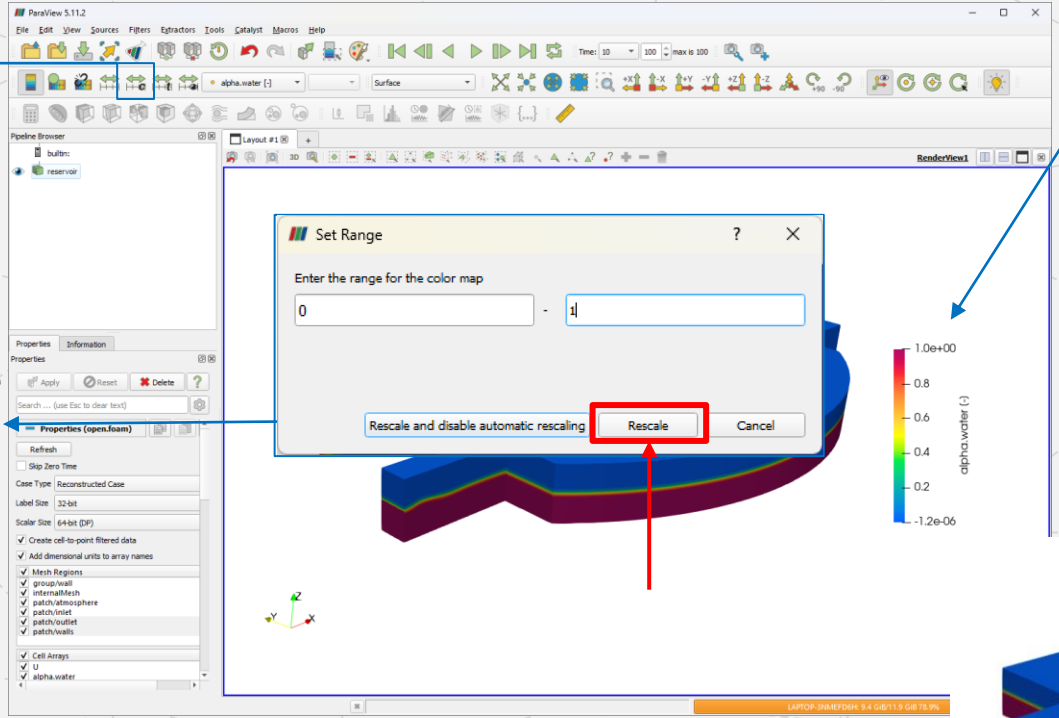
Adjusting colors, data range and labels

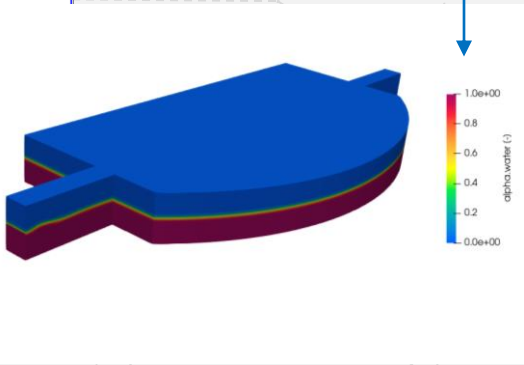
1 – click on Rescale to custom data range

2 – define range from 0 to 1 and click Rescale

Previous data range

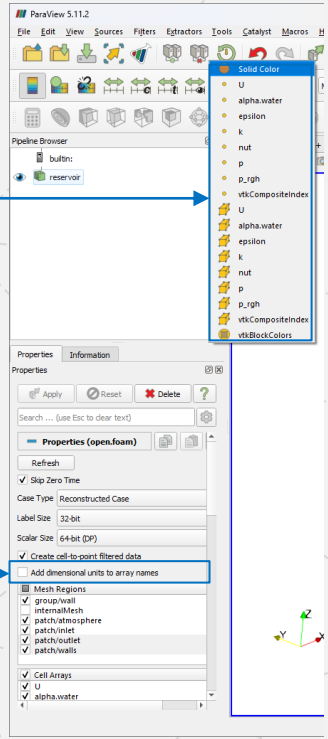
3 – New data range





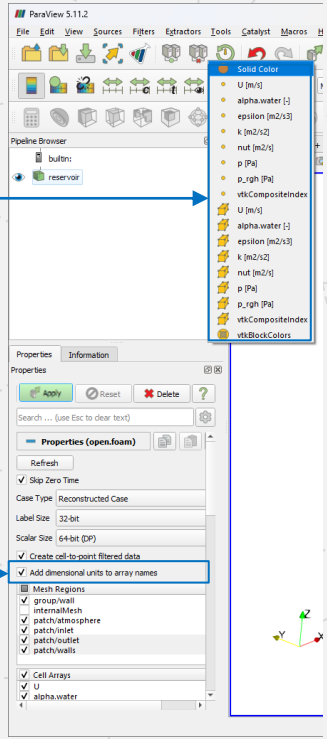
Displaying scalar and vector fields

Fields without units



Keep it unchecked

Fields with units

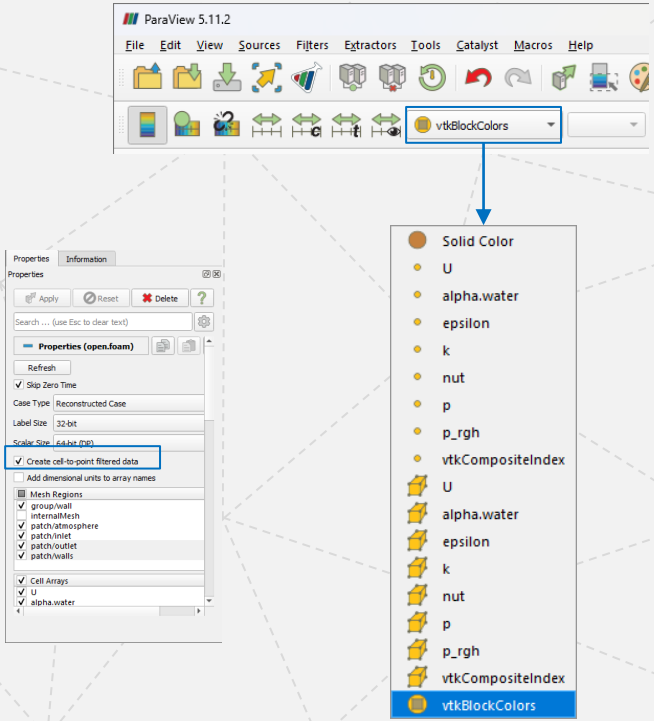


Check and click

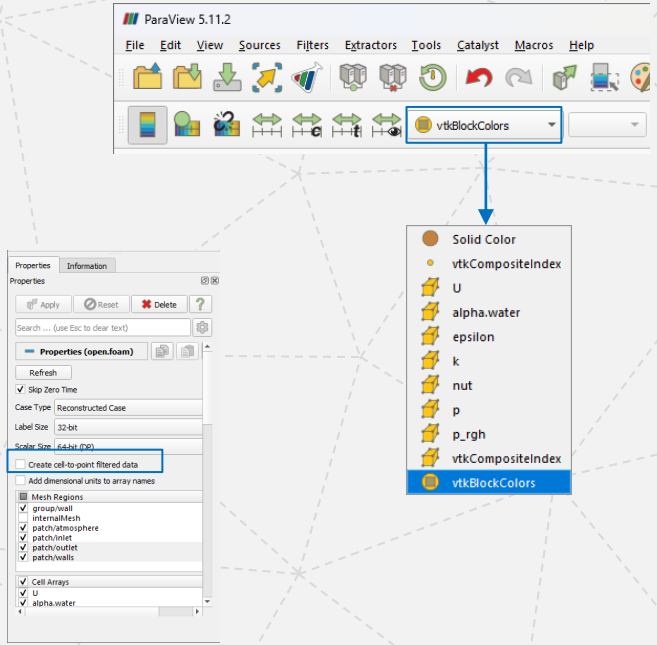
Displaying scalar and vector fields

By keeping the “Create cell-to-point filtered data” checked, fields with interpolated data between cells are available

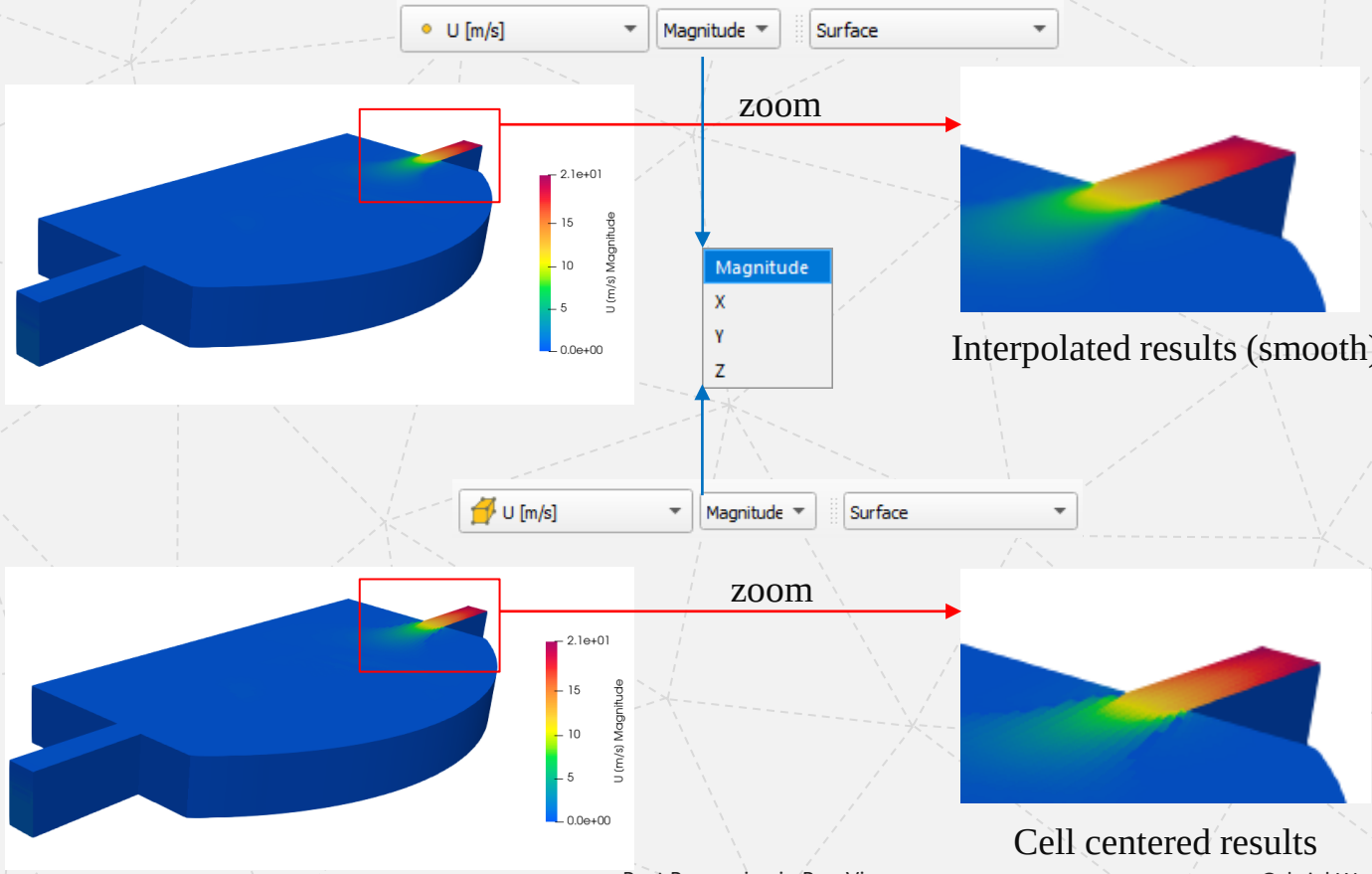
Point Data - interpolated



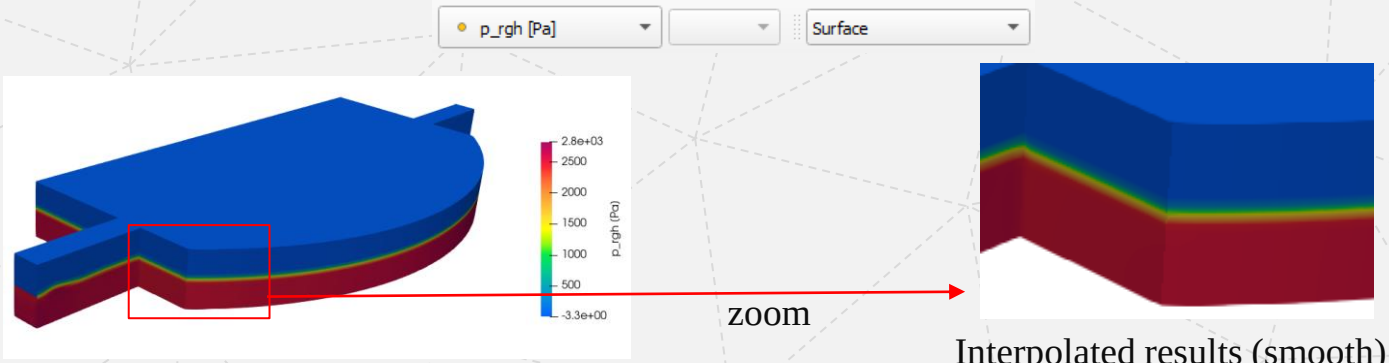
Cell Data



Displaying scalar and vector fields



Displaying scalar and vector fields

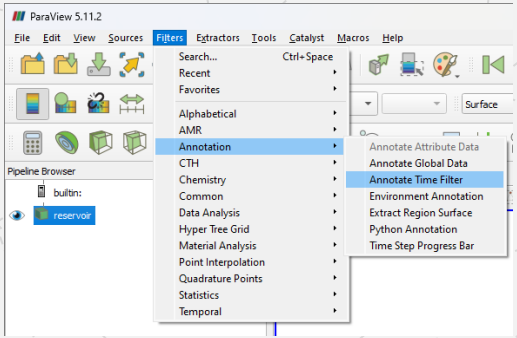


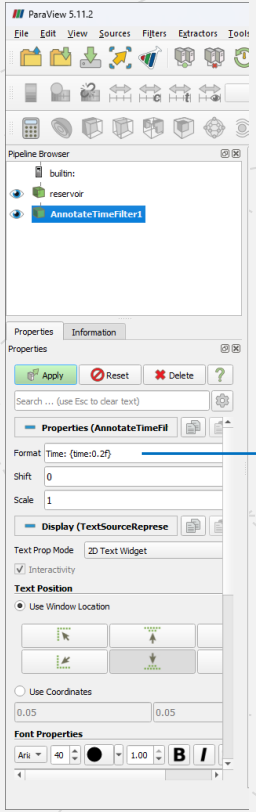
Applying filters

Annotate time filter

- Select the reservoir object in the pipeline browser
- Select the Annotate Time Filter in the menu bar

Filters > Annotation > Annotate Time Filter





Define format as
"Time: {time:0.2f}

0.2 means it wil contain 2 decimal numbers

Location in Canvas

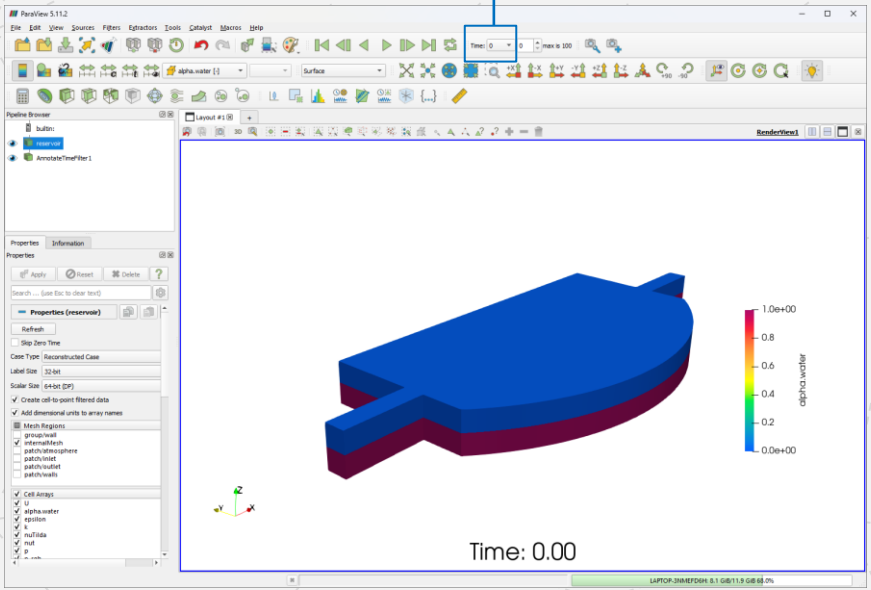
Define font properties

Applying filters

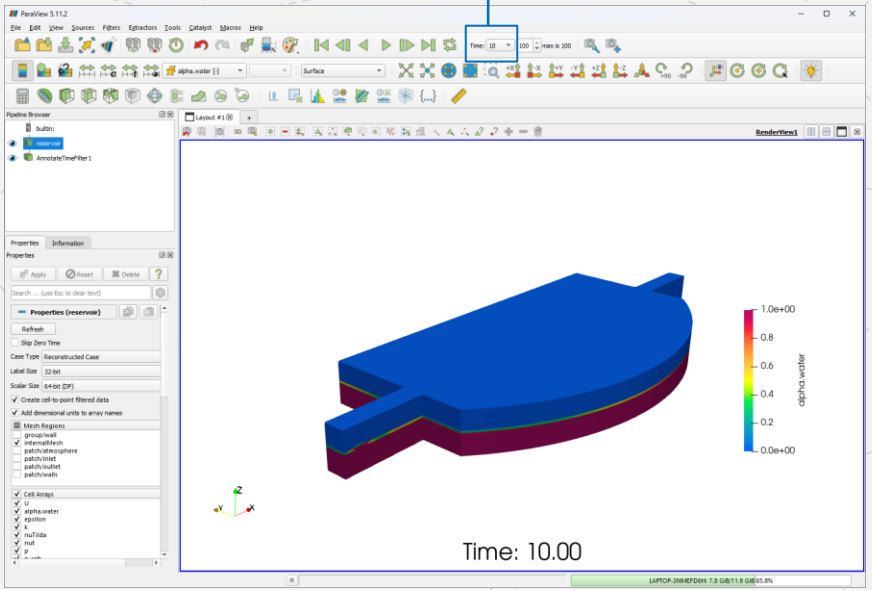
Annotate time filter

Time will be displayed in the canvas according to the instante of time being visualized

0



10

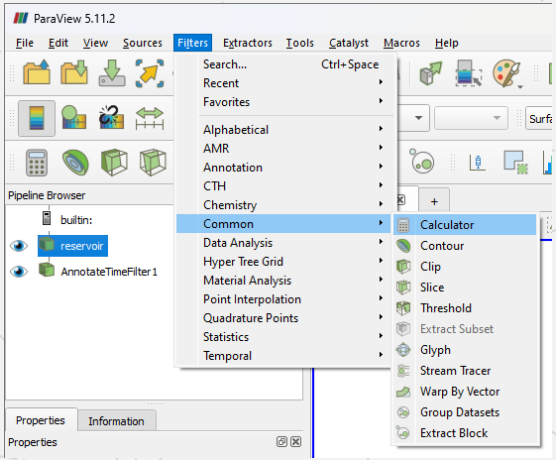


Applying filters

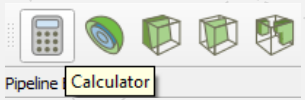
- Select the reservoir object in the pipeline browser

- Select the Calculator filter in the menu bar

Filters > Common > Calculator

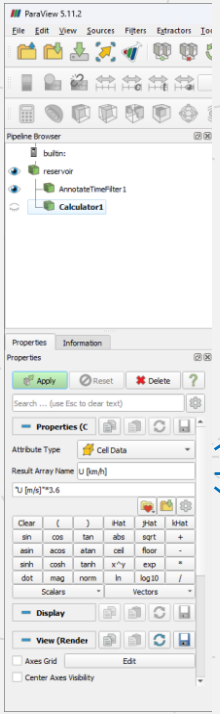


Or select the object and click in the calculator filter above Pipeline Browser



Calculator

Creates an variable field according to an expression



Define if it is point data or cell data

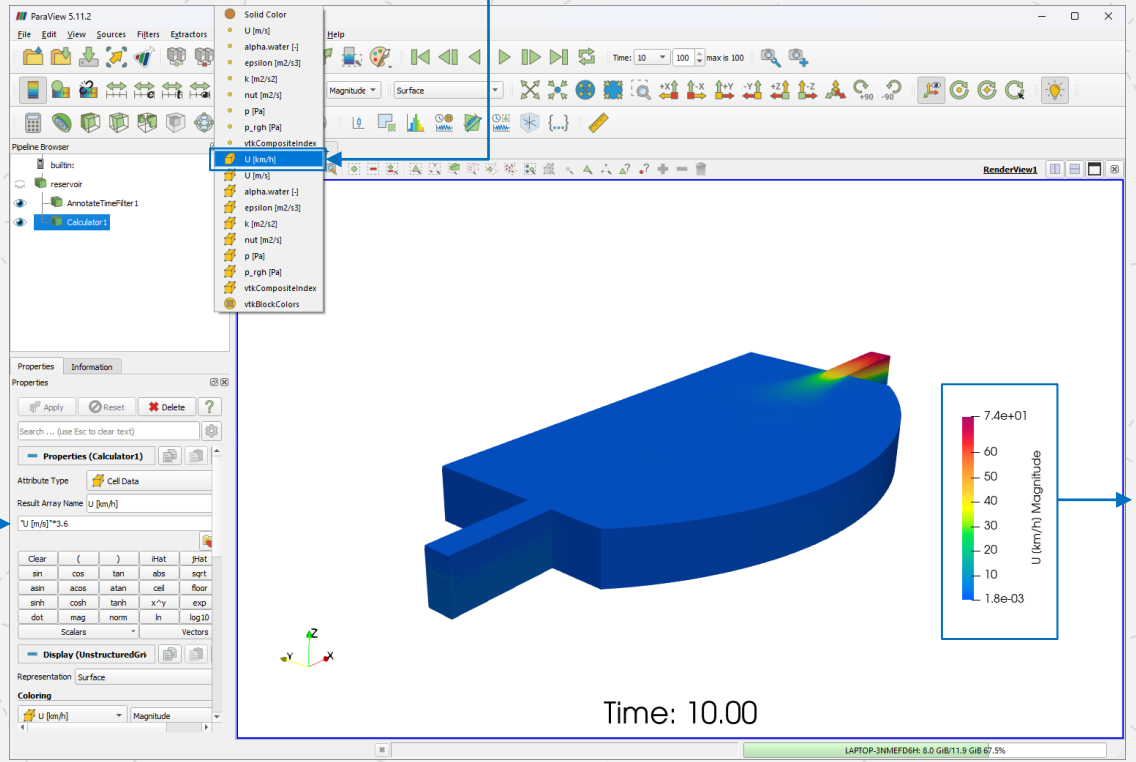
Name – U [km/h]

Equation

Applying filters

Calculator

Created variable – U [km/h]



"U[m/s]"*3.6

Data range in km/h

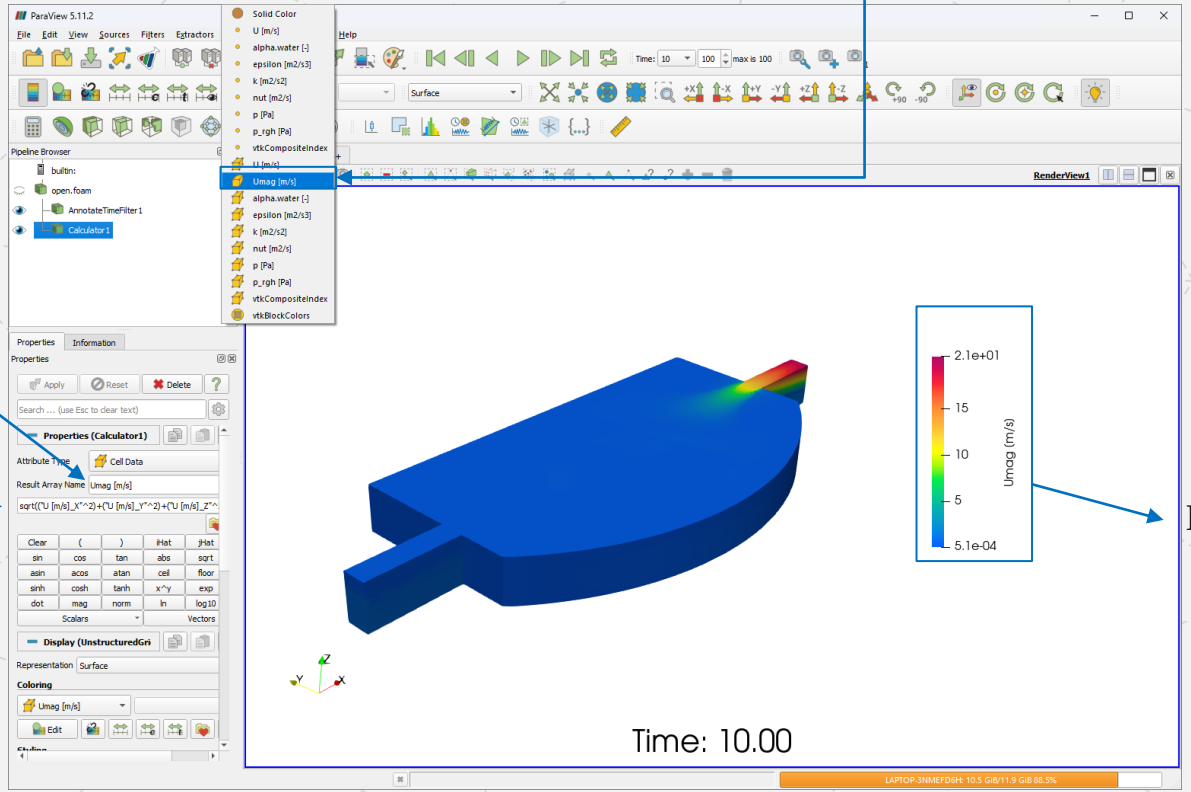
Applying filters

Calculator

Created variable calculating the magnitude of velocity – Umag [m/s]

Umag [m/s]

sqrt((“U [m/s]_X”^2)+(“U [m/s]_Y”^2)+(“U [m/s]_Z”^2))



Data range in m/s

CRheo@IPC

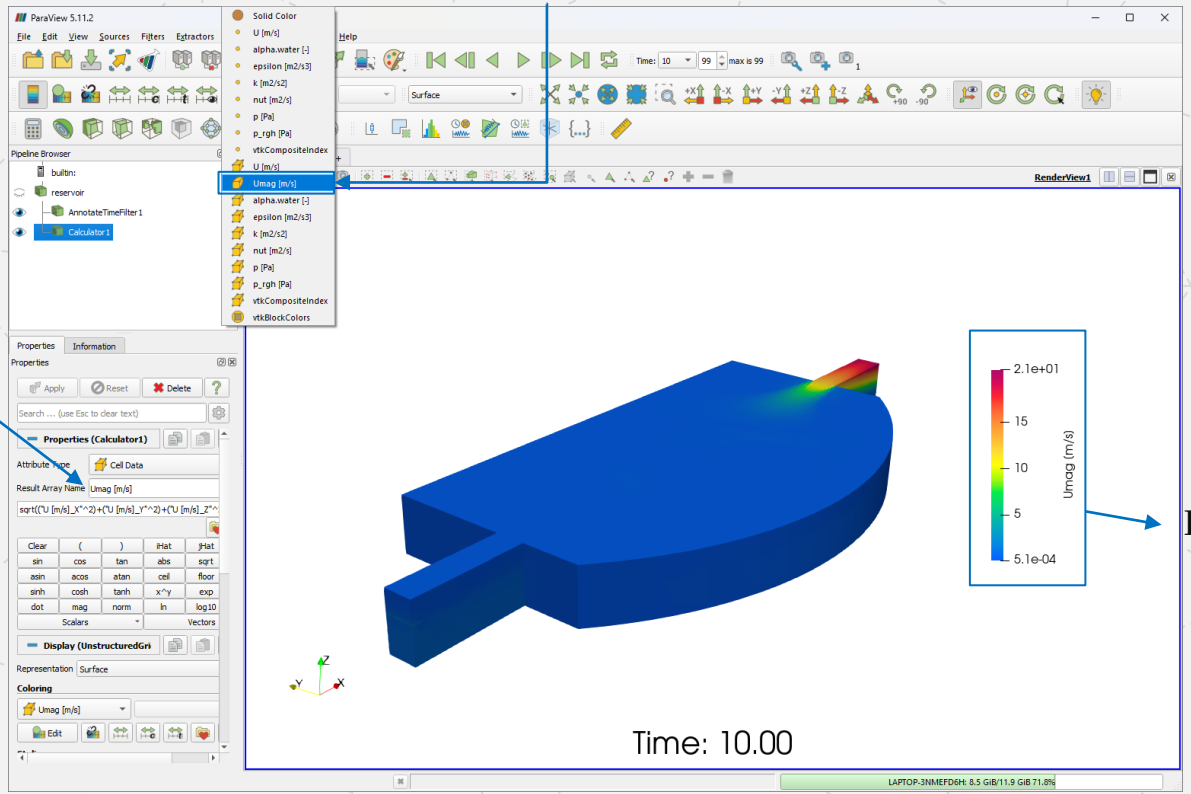
Applying filters

Calculator

Created variable calculating the magnitude of velocity – Umag [m/s] – it is the same as field U [m/s] magnitude

Umag [m/s]

sqrt((“U [m/s]_X”^2)+(“U [m/s]_X”^2)+(“U [m/s]_X”^2))



Data range in m/s

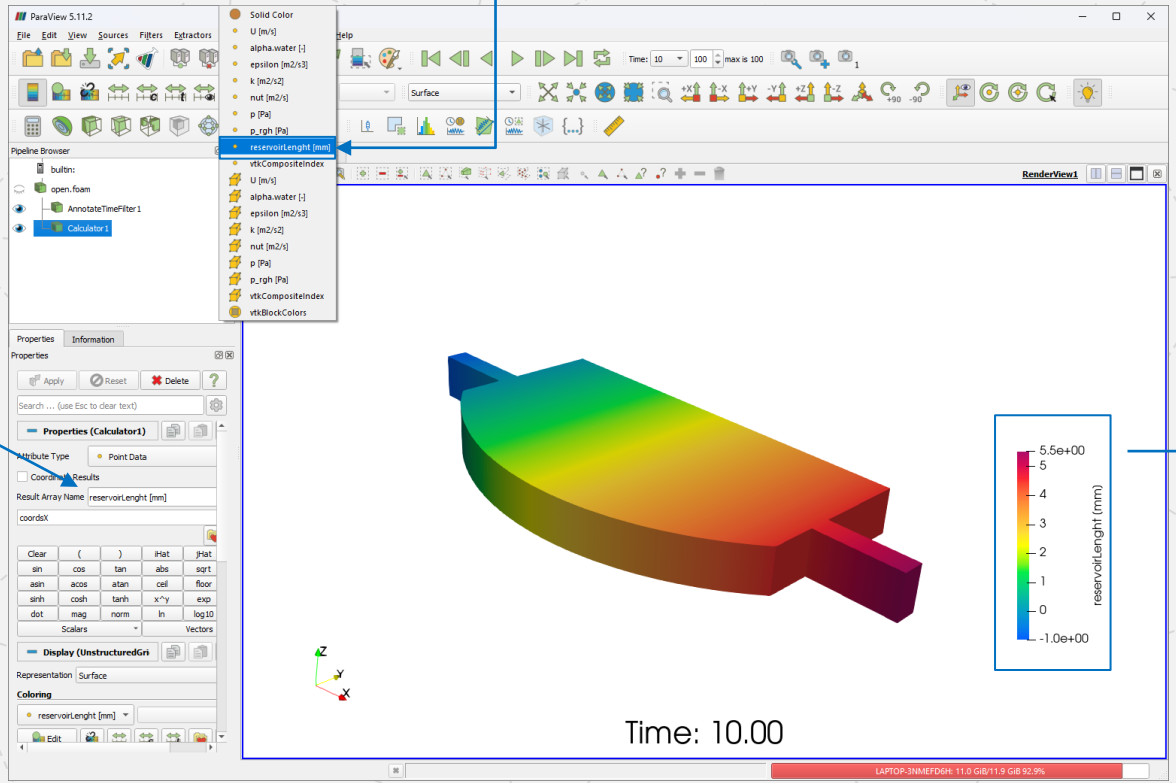
Applying filters

Calculator

Created variable – reservoirLenght [mm]

reservoirLenght [mm]

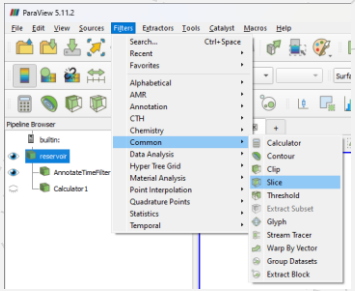
coordsX



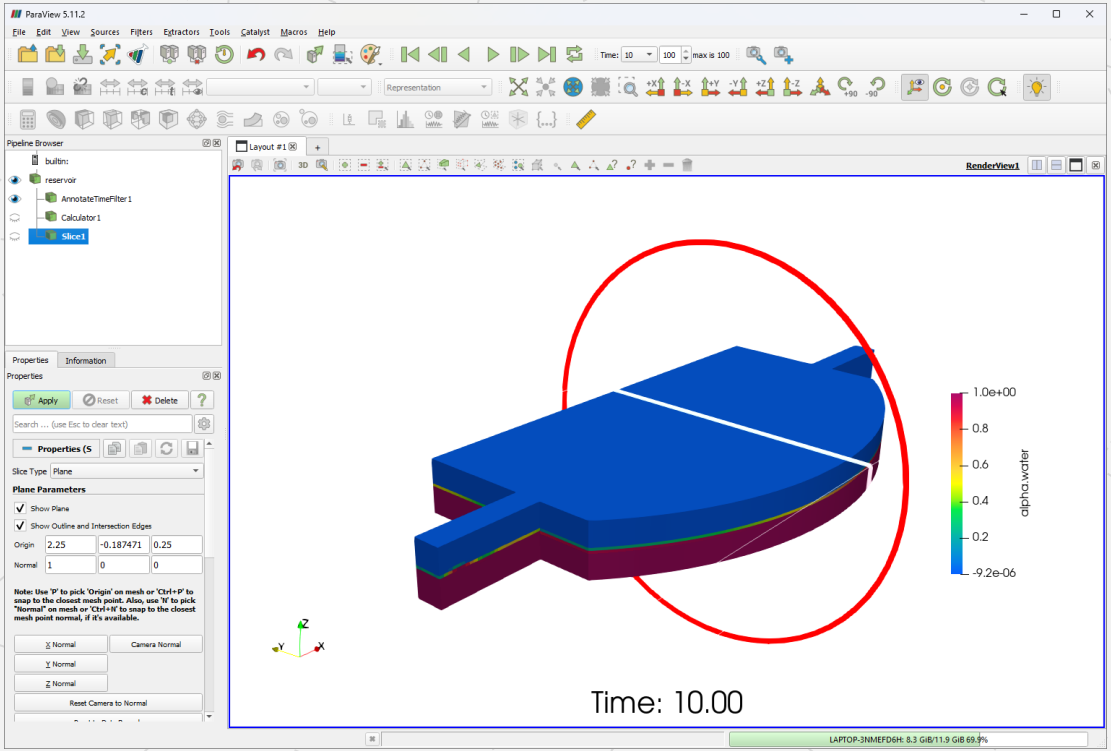
Data range in mm

Applying filters

Slice



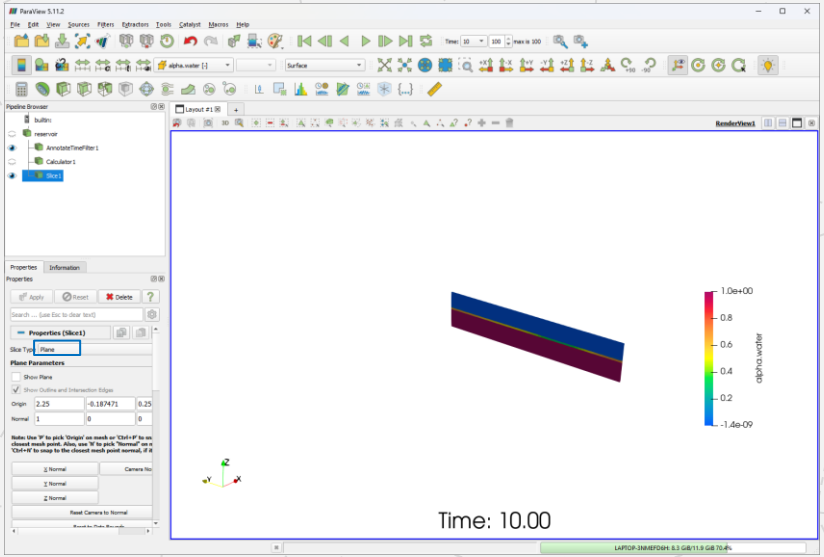
Slices the object in a Box, Cylinder, Plane or Sphere



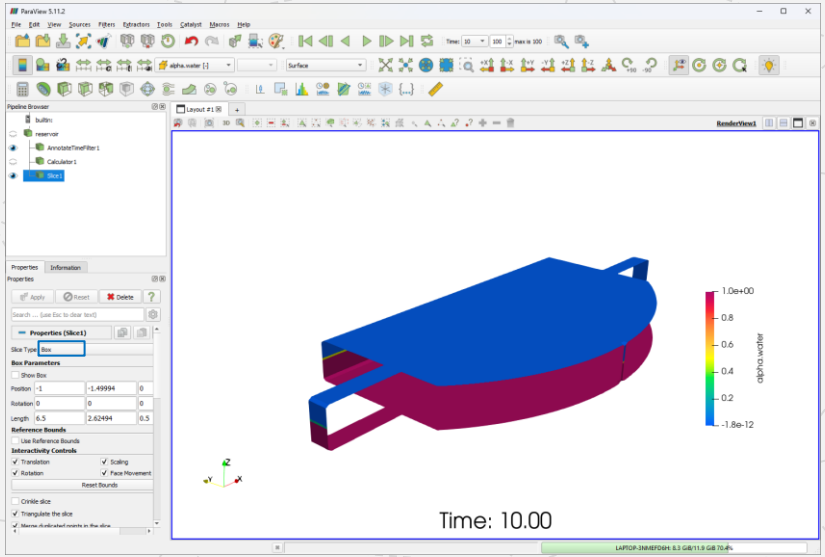
Applying filters

Slice

Slice - Plane



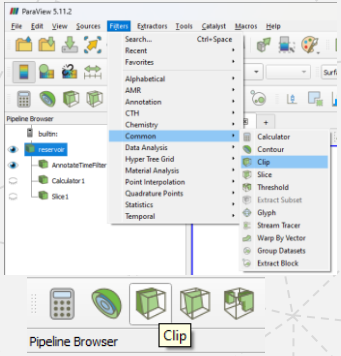
Slice - Box

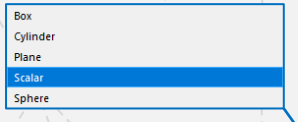


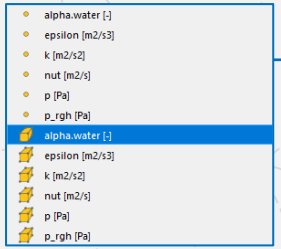
Applying filters

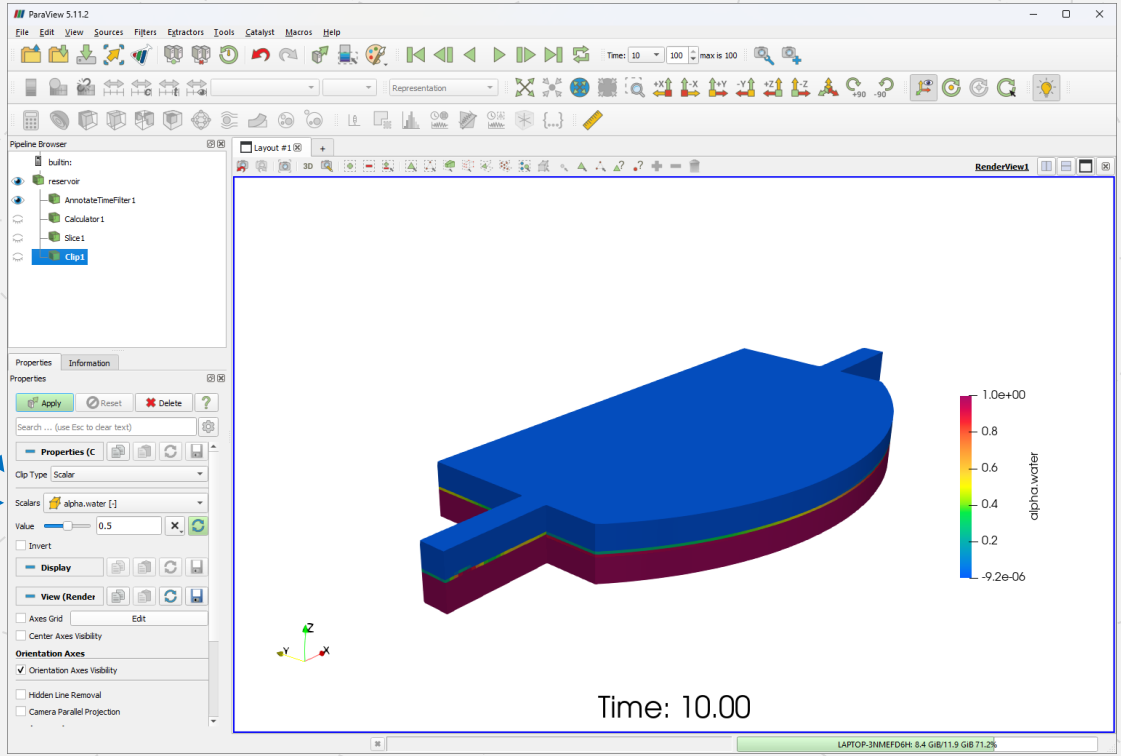
Clip

Clips the object's field greater or lower a defined value







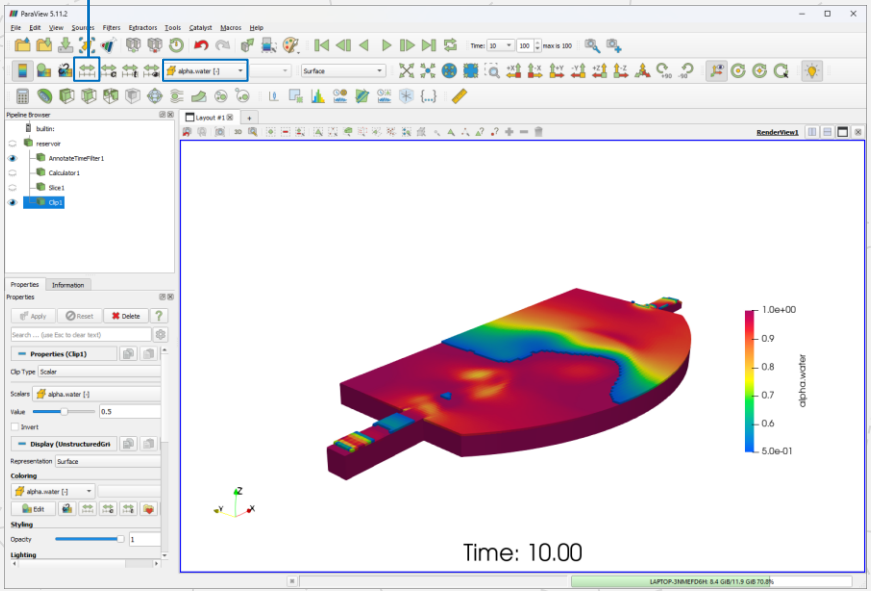


Applying filters

Clip

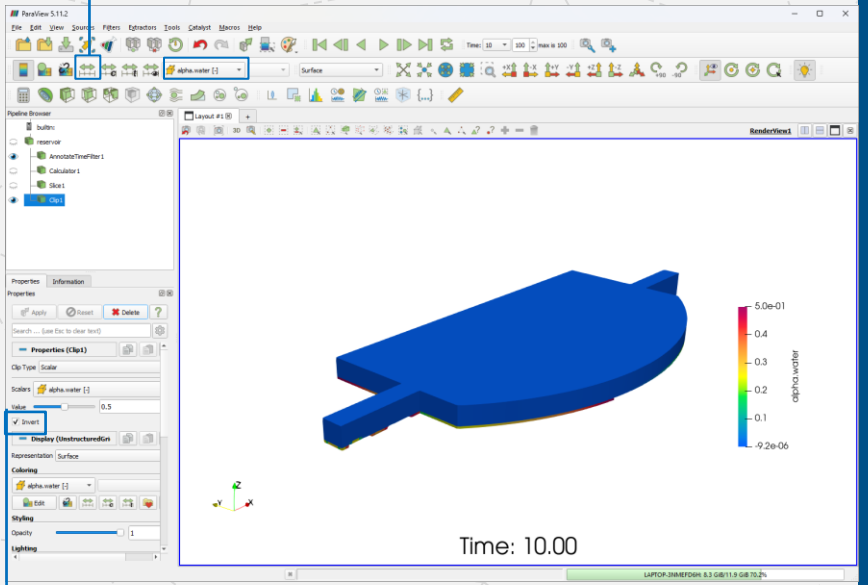
Click Rescale to Data Range

Clip – alpha.water greater than 0.5



Click Rescale to Data Range

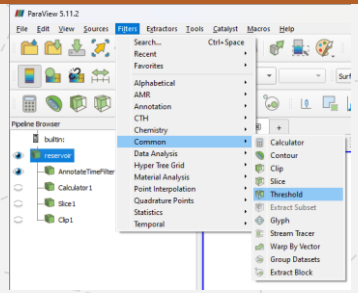
Clip – alpha.water smaller than 0.5



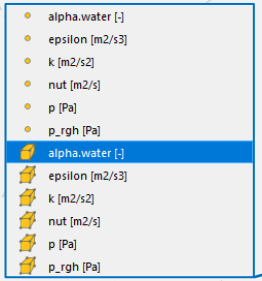
Check “Invert”

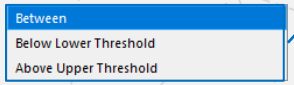
Applying filters

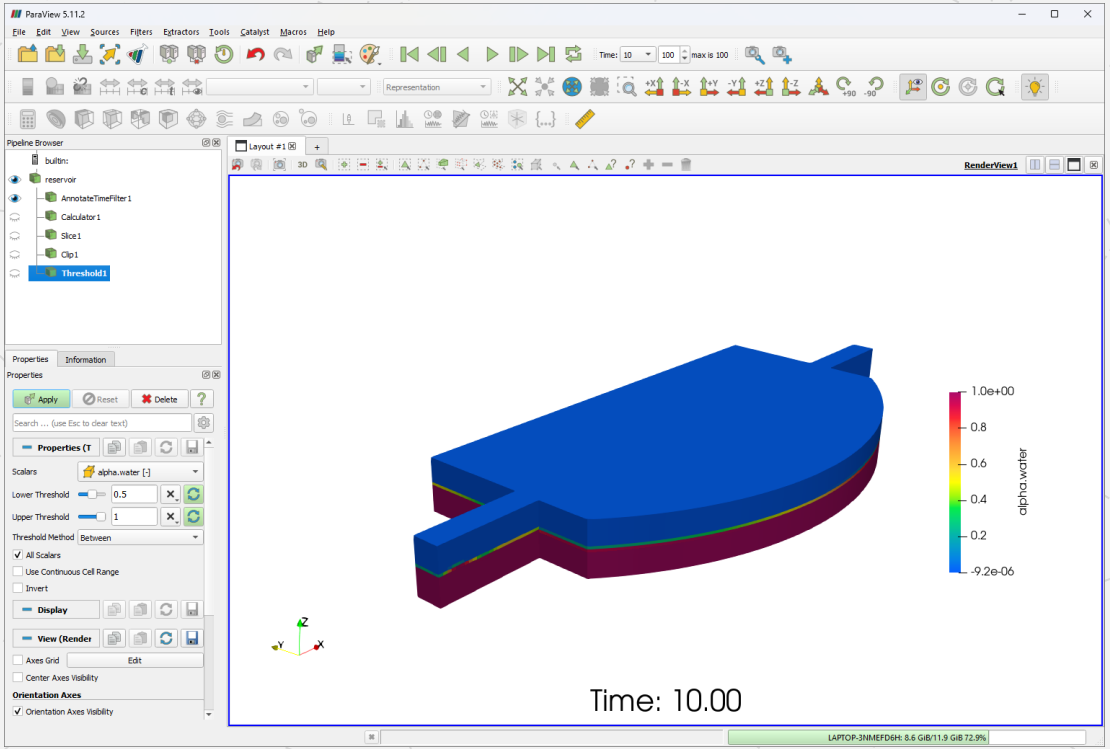
Threshold



Thresholds object's scalar field between, below or above defined value







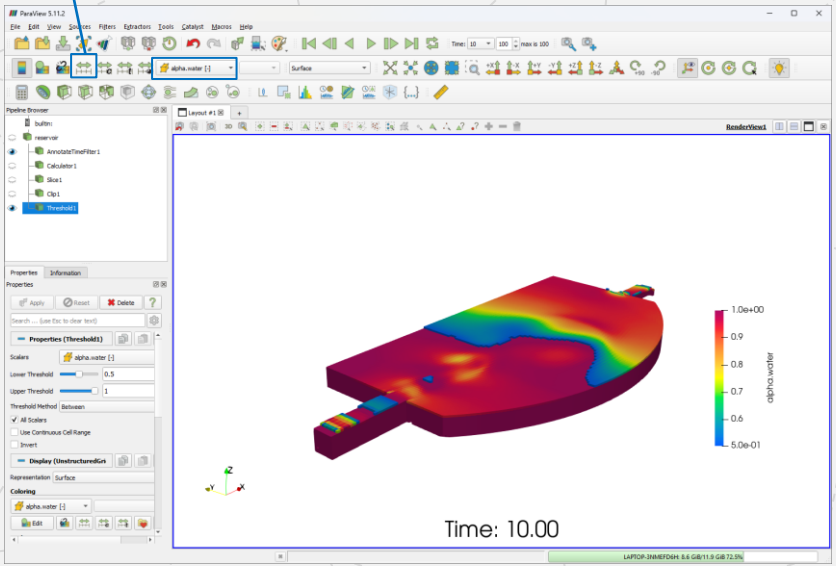
CRheo@IPC

Applying filters

Threshold

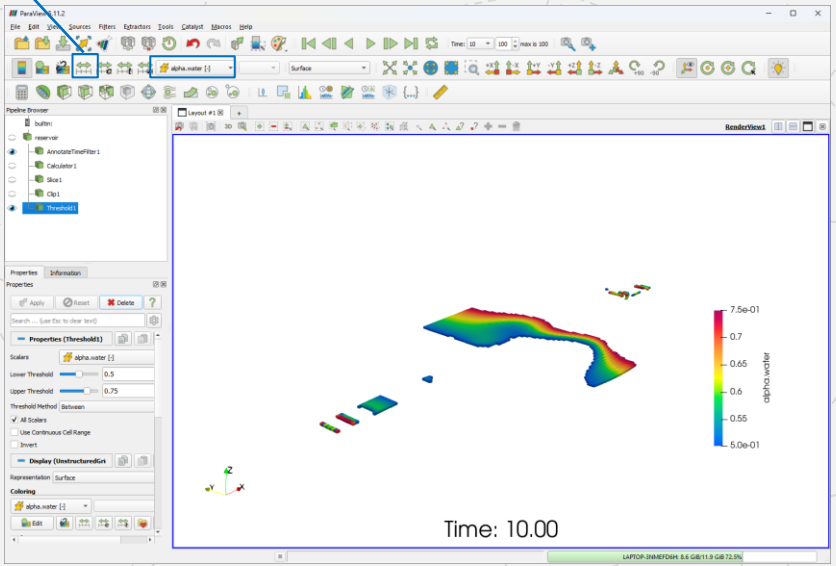
Click Rescale to Data Range

Threshold – alpha.water between 0.5 and 1



Click Rescale to Data Range

Threshold – alpha.water between 0.5 and 0.75



This is similar to clip > 0.5, since alpha.water values varies from 0 to 1

Applying filters

Threshold

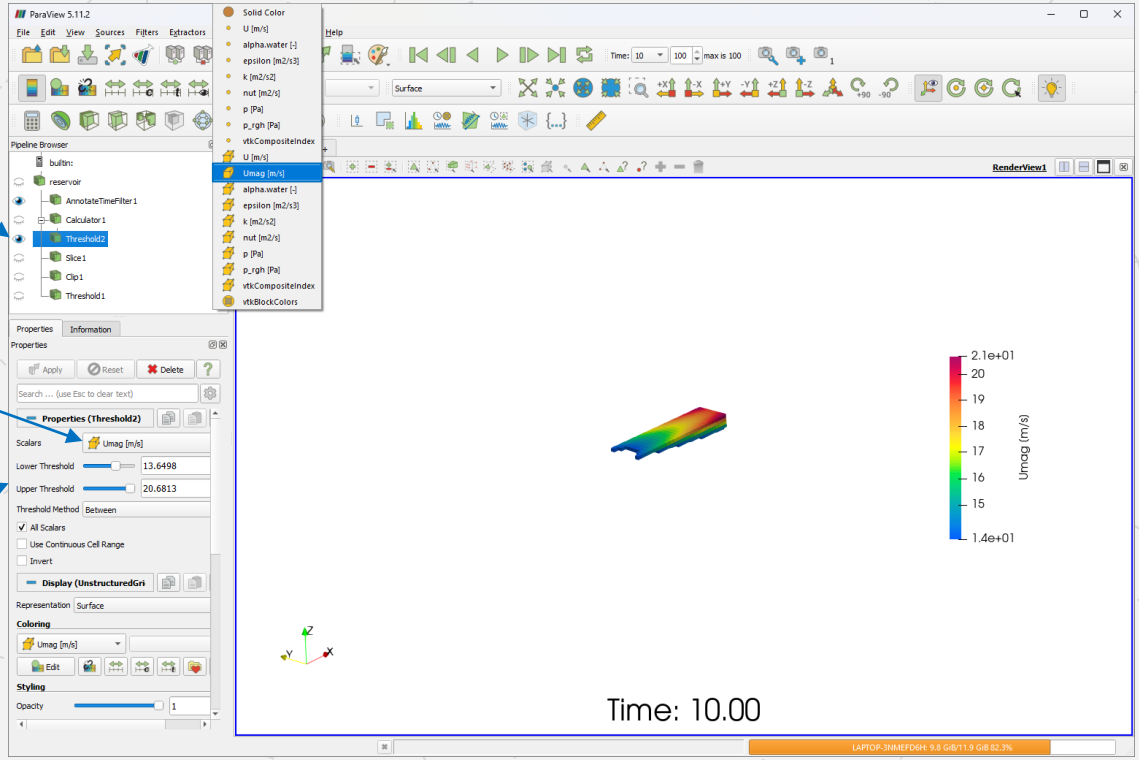
- Select the “calculator1” filter in the pipeline browser

- Select the Threshold filter in the menu bar

Define threshold to scalar Umag [m/s]

Define threshold range

Threshold to calculated Umag scalar field

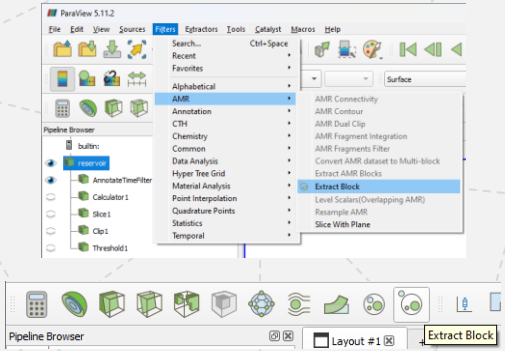


Applying filters

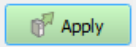
Extract block

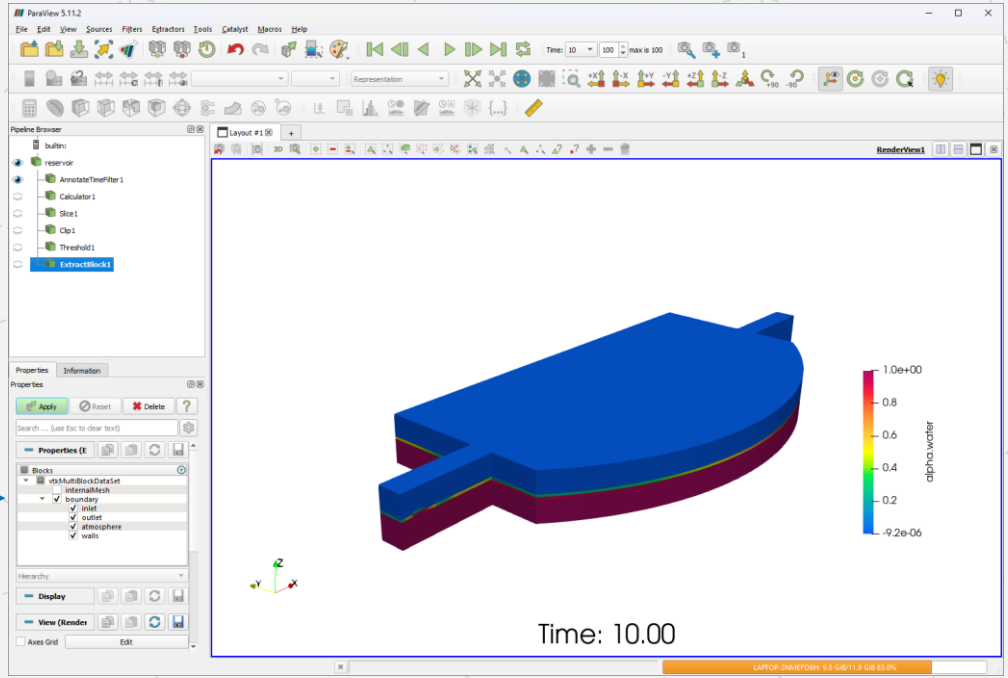
Extracting blocks allows you to obtain different objects from the same object.

It is particularly helpfull to analyse surface, and if you want to apply filter to the internal mesh and visualize surfaces.



Select boundary and click



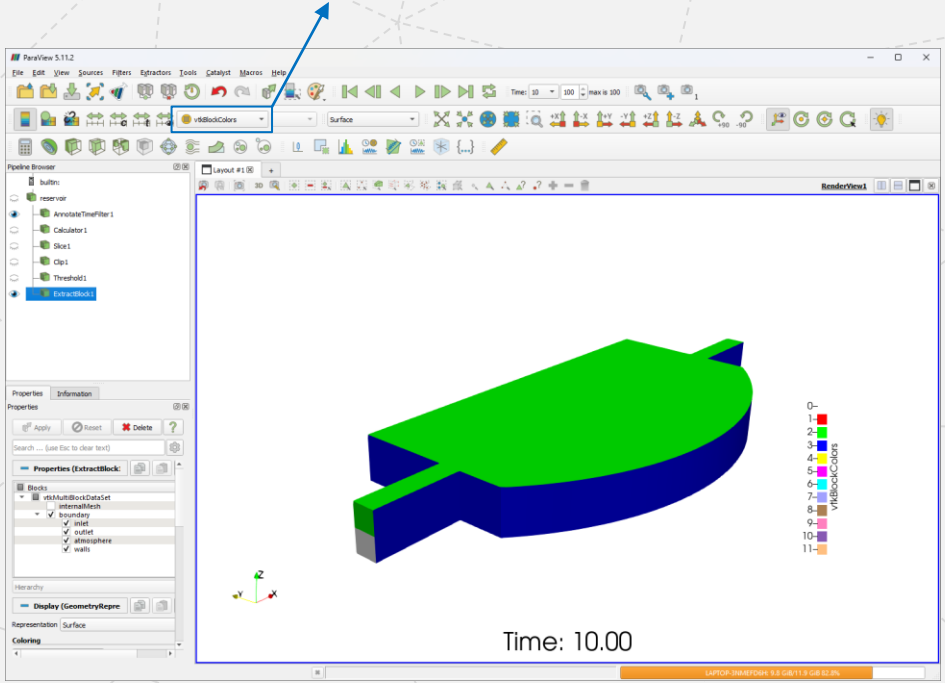


Applying filters

Extract block

By that, you have created ExtractBlock1 by extracting all the surfaces of your object.

Select coloring *vtkBlockColors* to visualize.



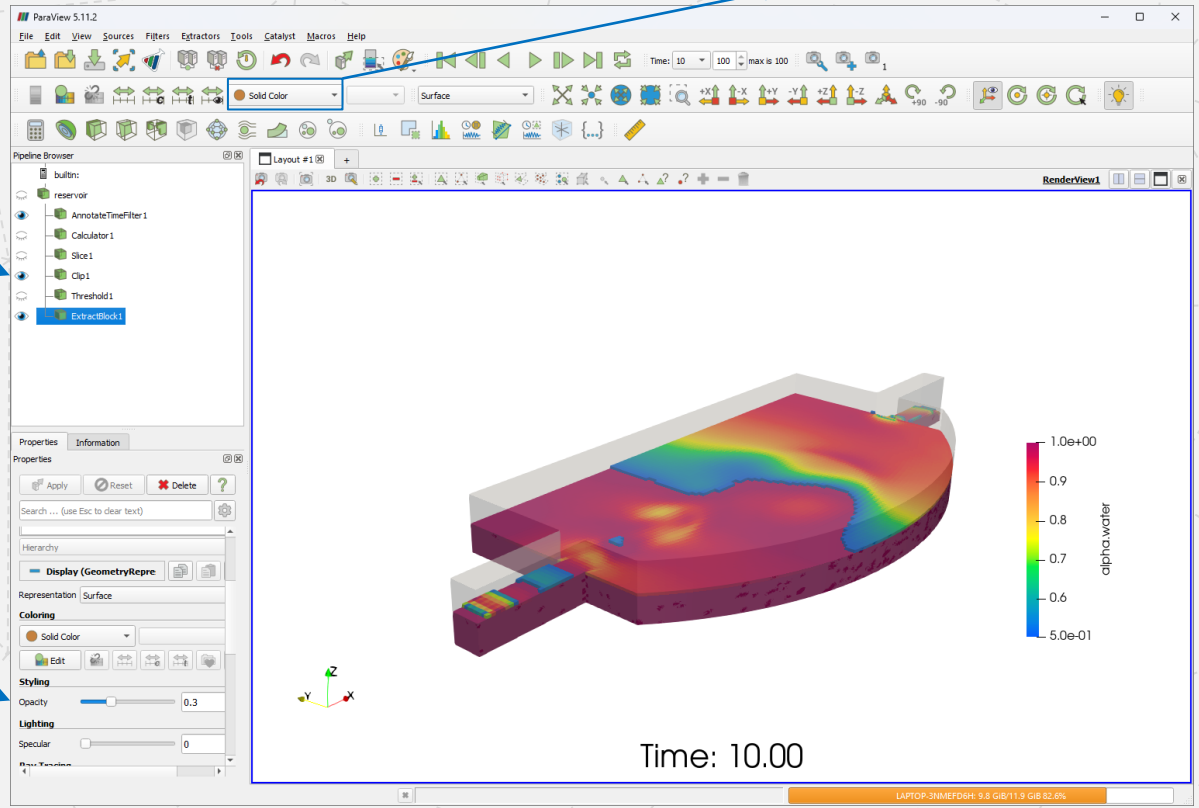
Applying filters

Extract block

Select ExtractBlock1 coloring to *Solid Color*

Enable Clip1 visualization

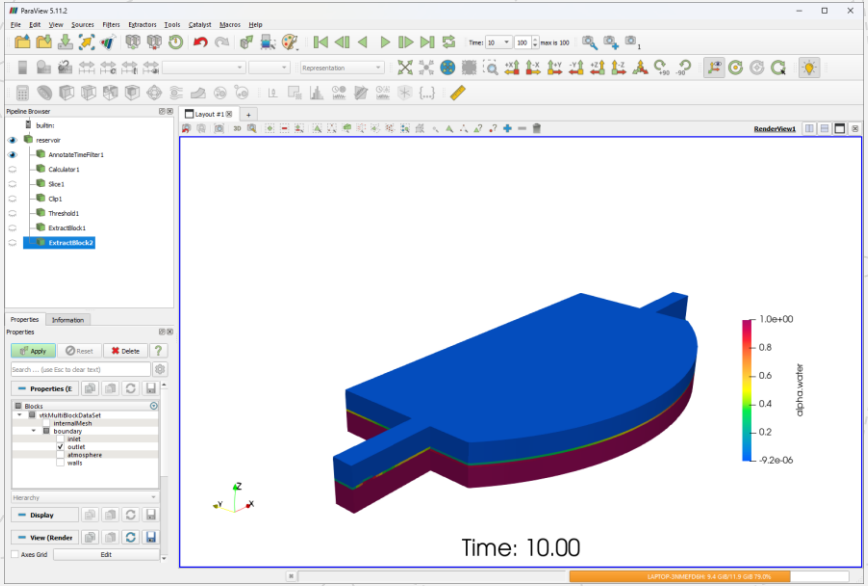
Set ExtractBlock1 opacity to 0.3

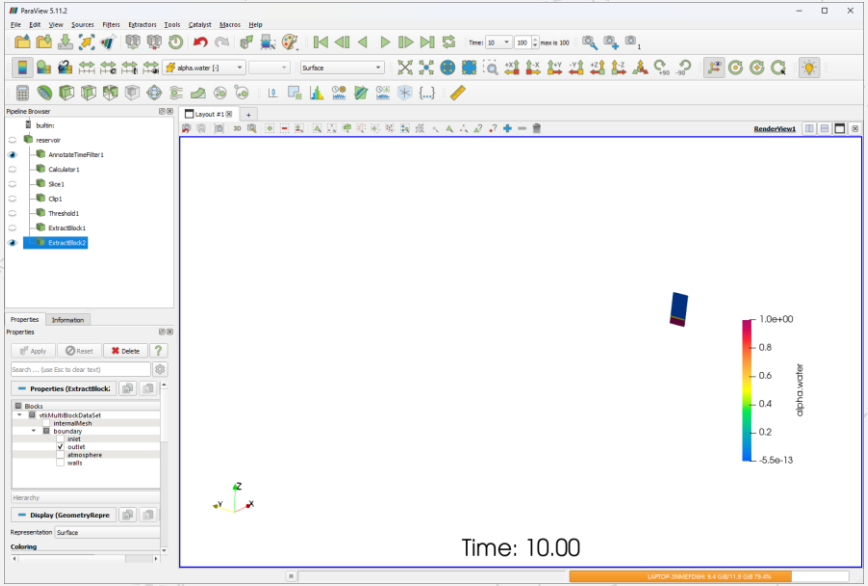


Applying filters

Extract block

Create new extract block just with the Outlet patch to analize it.

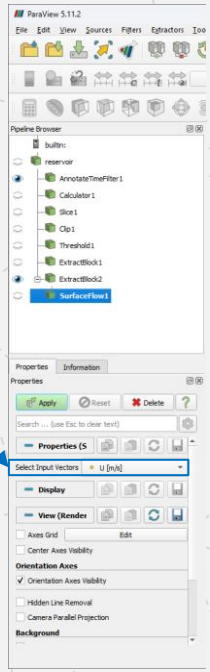




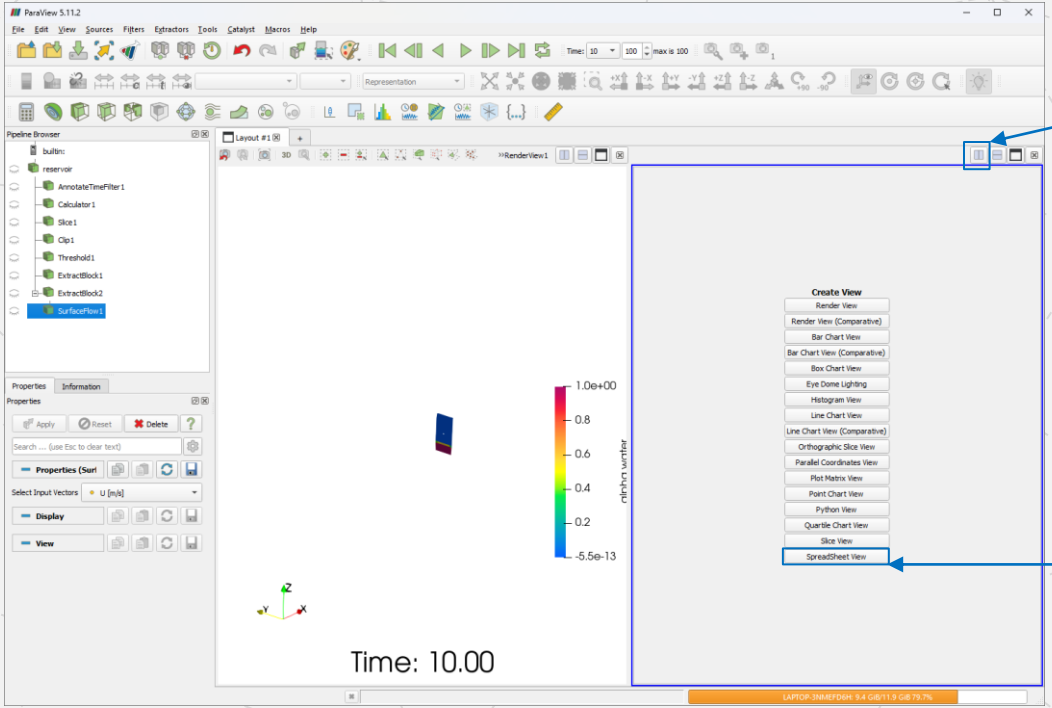
Applying filters

Extract block – surface flow

Use the *Surface Flow* filter in the ExtractBlock2 and open it in a SpreadSheet view



U [m/s]



1 – split view in horizontal

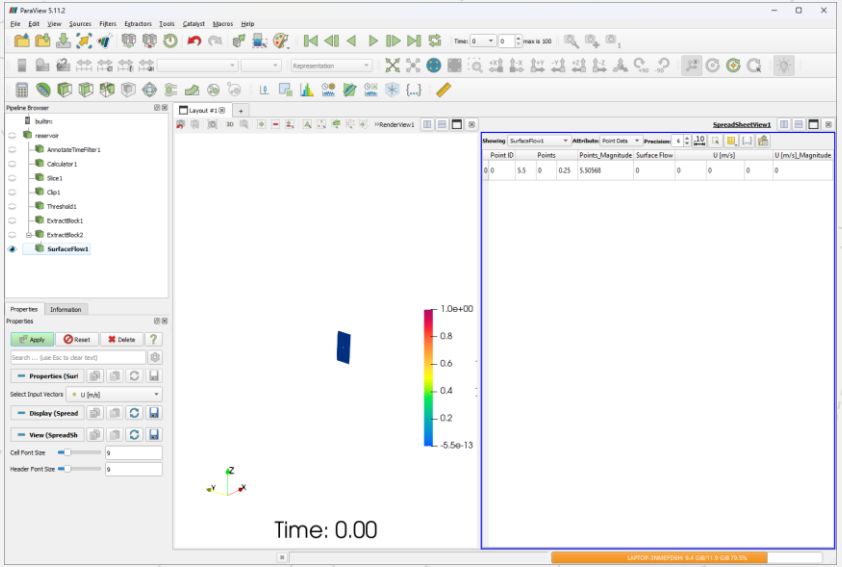
2 – select SpreadSheet View

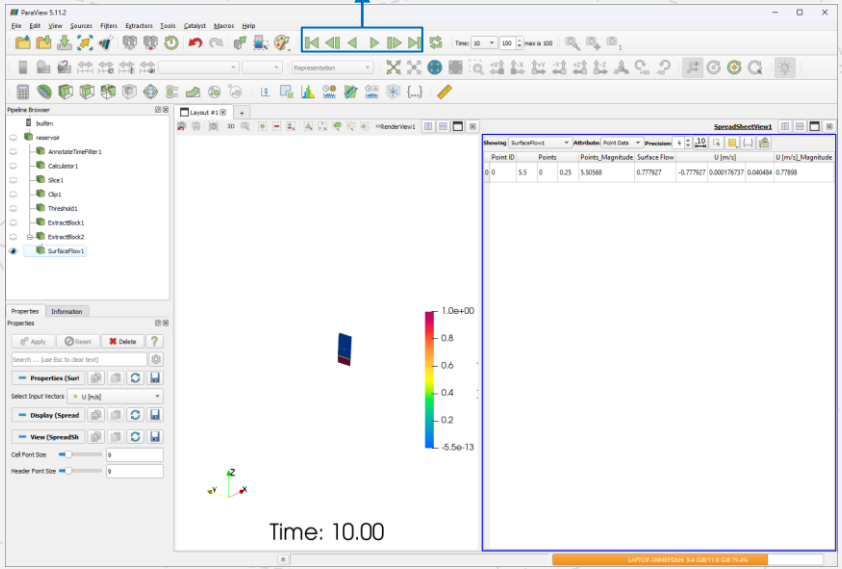
Applying filters

Extract block – surface flow

With that, you can obtain Surface flow and velocity data for the current instant of time.

Move forward or backward in time to visualize data for all instants of time



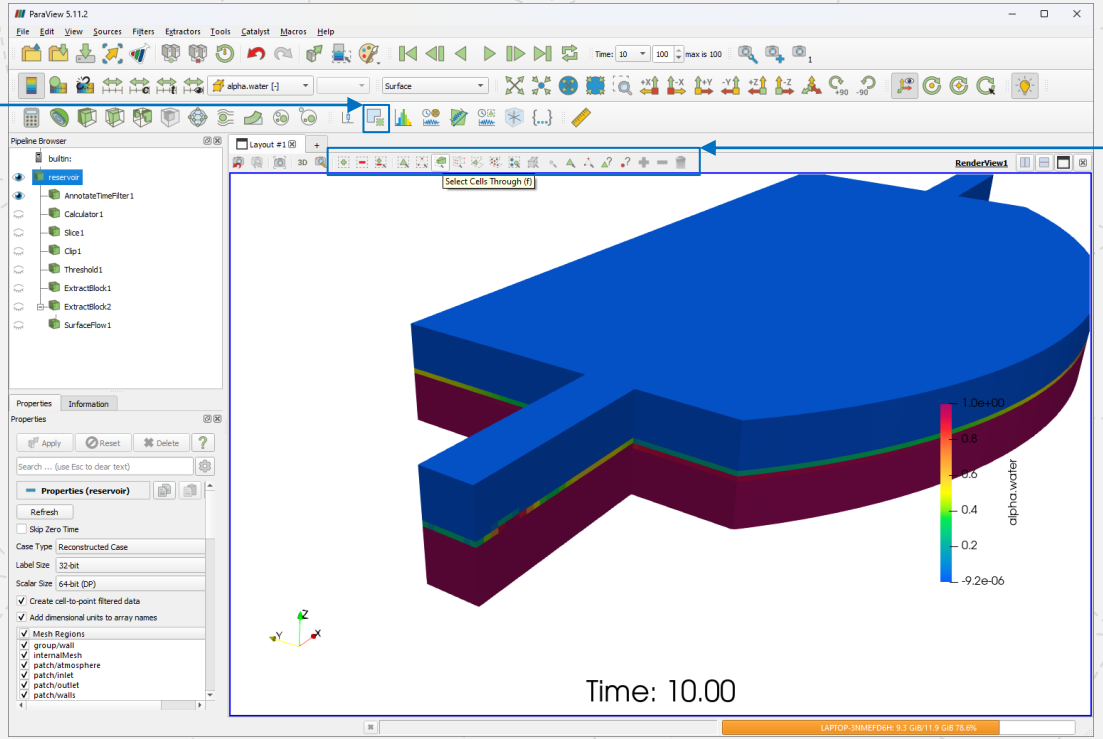


Applying filters

Extract selection

Extract Selection allow you to extract regions and analyse them independently. Use selection methods to identify the desired region, and apply the *Extract Selection* filter.

Extract Selection filter



Selection methods

CRheo@IPC

Applying filters

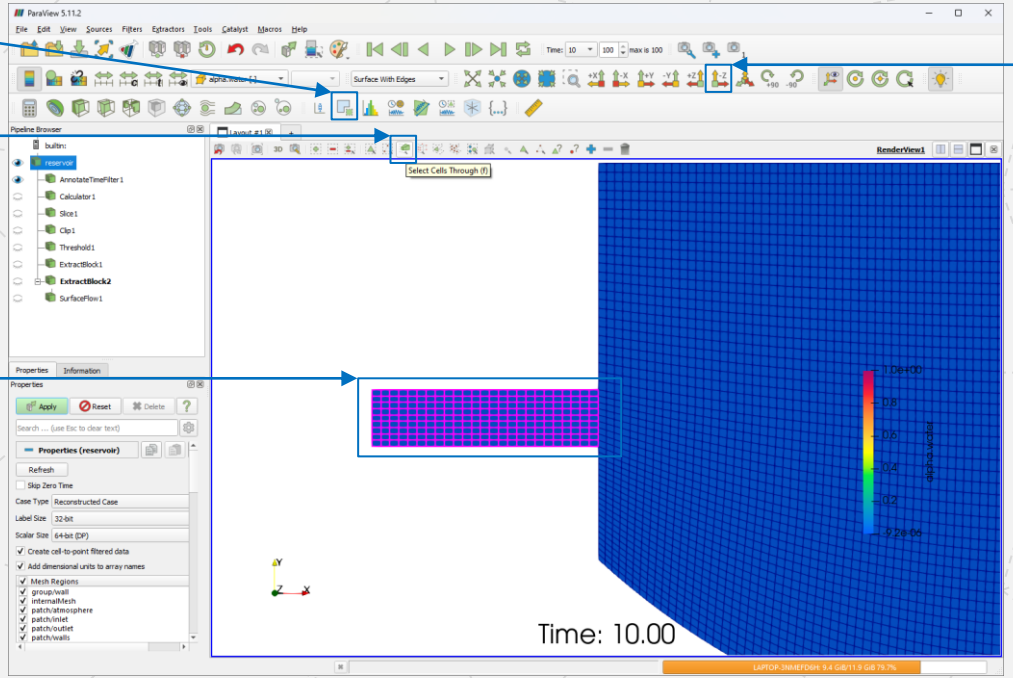
Extract selection

Extract Selection allow you to extract regions and analyse them independently. Use selection methods to identify the desired region, and apply the *Extract Selection* filter.

4 - Extract Selection

2 - Use “Select Cells Through”

3 - Select this region

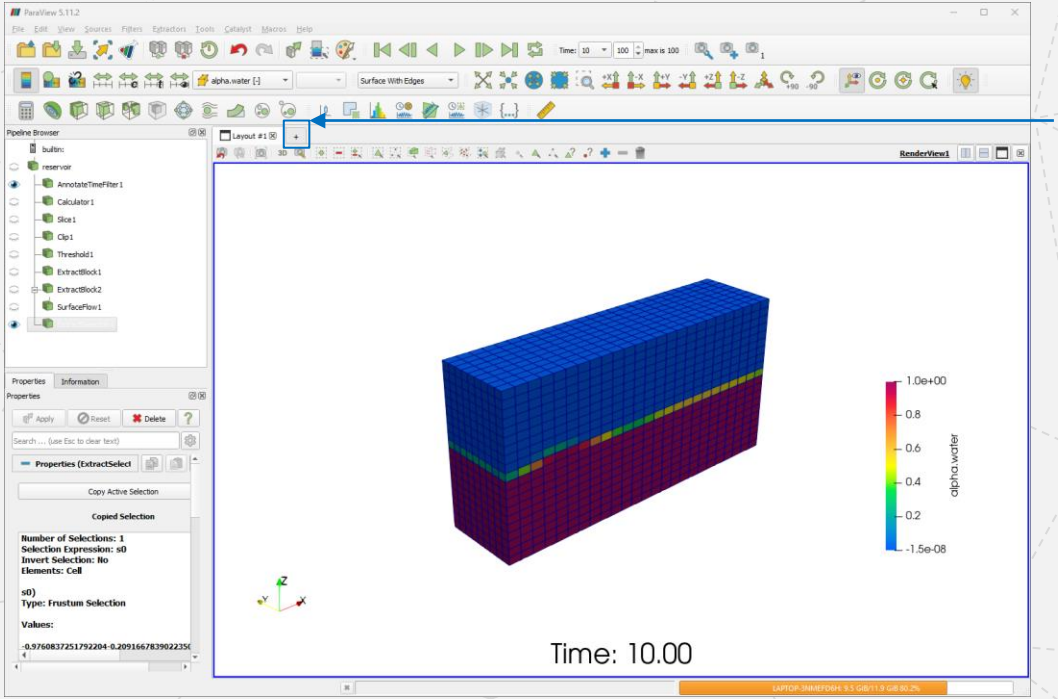


1 - Change view

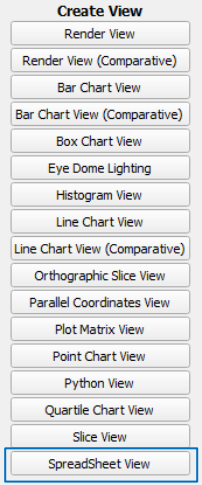
Applying filters

Extract selection

Extracted selection



Open a new tab and select SpreadSheet View



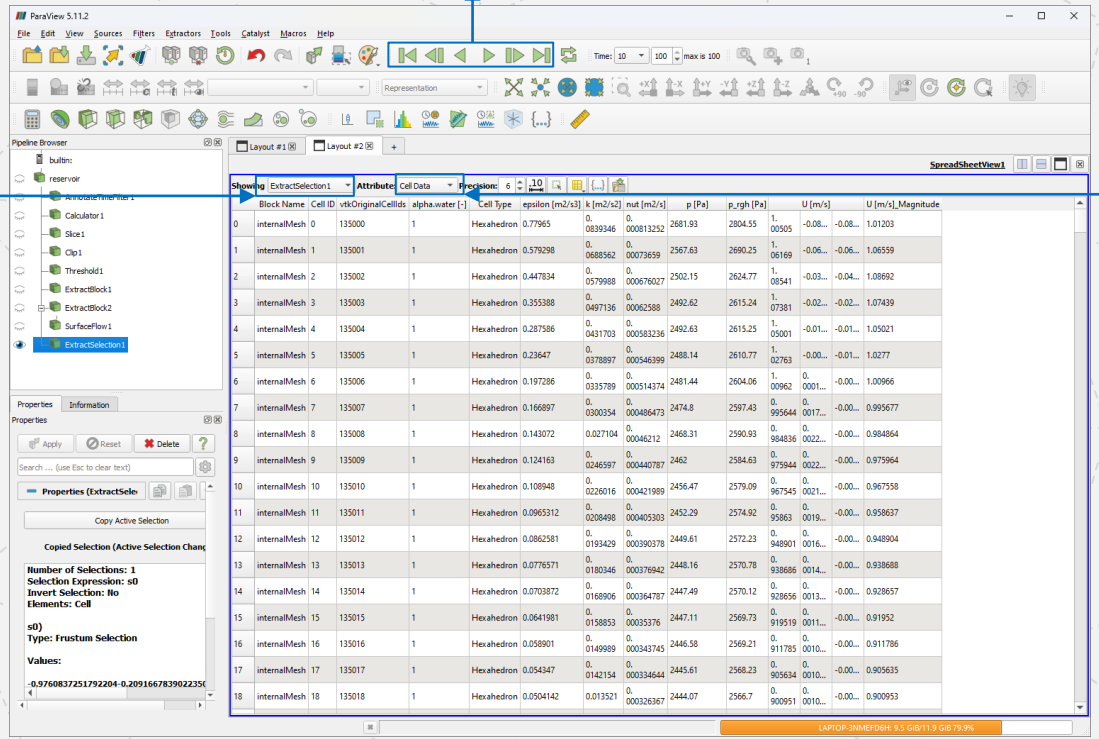
Applying filters

Extract selection

Information of all the cells in the extracted selection is displayed for the current instant of time.

Move forward or backward in time to visualize data for all instants of time

ExtractSelection1



Cell data

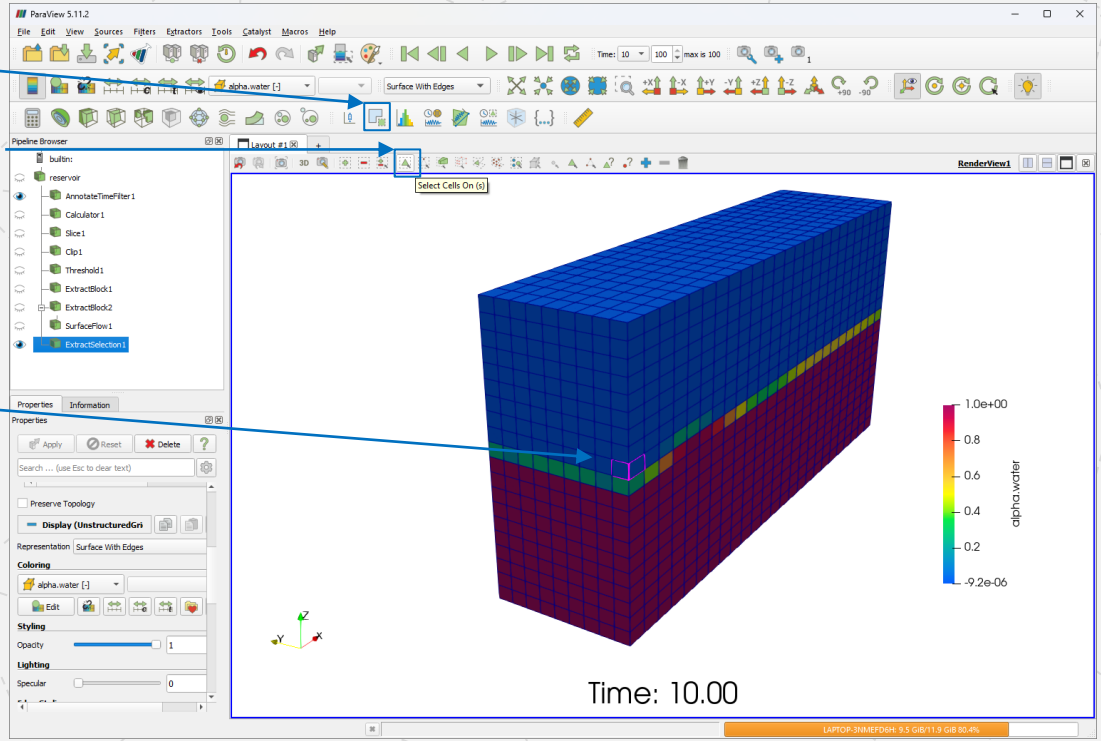
Select cells and show values

Similarly to *Extract Selection* filter, *Select cells on* allows to analyse data for a given cell (not region).

3 - Extract Selection

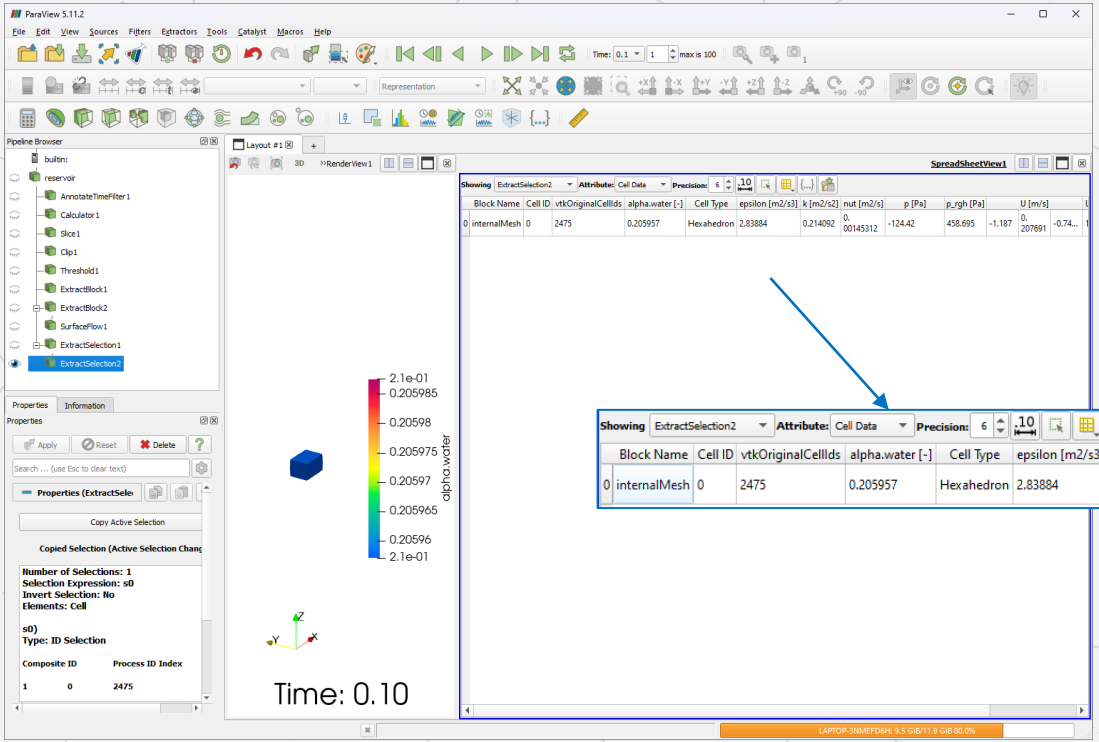
1 - Use “Select Cells on” to select a cell

3 - Select cell



Select cells and show values

Split horizontal view and select SpreadSheet View. Information of the cell will be displayed.



Showing ExtractSelection2 Attribute: Cell Data Precision: 6

	Block Name	Cell ID	vtkOriginalCellIds	alpha.water [-]	Cell Type	epsilon [m2/s3]	k [m2/s2]	nut [m2/s]	p [Pa]	p_rgh [Pa]	U [m/s]		U [m/s]_Magnitude
0	internalMesh	0	2475	0.205957	Hexahedron	2.83884	0.214092	0. 00145312	-124.42	458.695	-1.187	0. 207691	-0.74...	1.41754

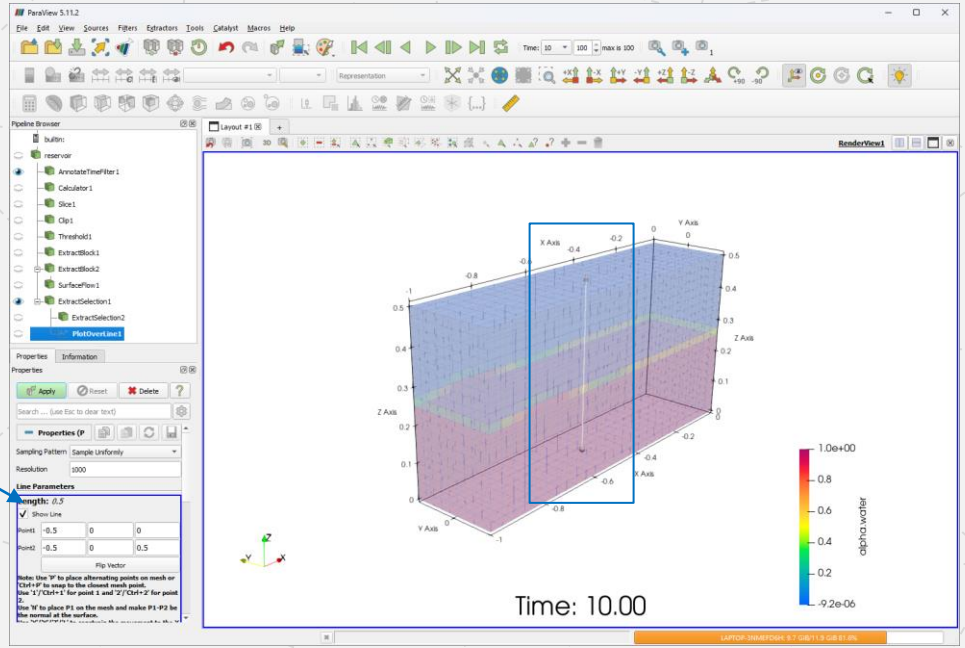
Plot over line

Select the ExtractSelection1 and apply the *Plot Over Line* filter.

Enter coordinates (-0.5 0 0) and (-0.5 0 0.5) in the properties panel. Change ExtractSelection1 opacity to view the line.

Check the “Axes grid” box in the properties panel of the ExtractSelection1 to view dimensions.

Lines coordinates

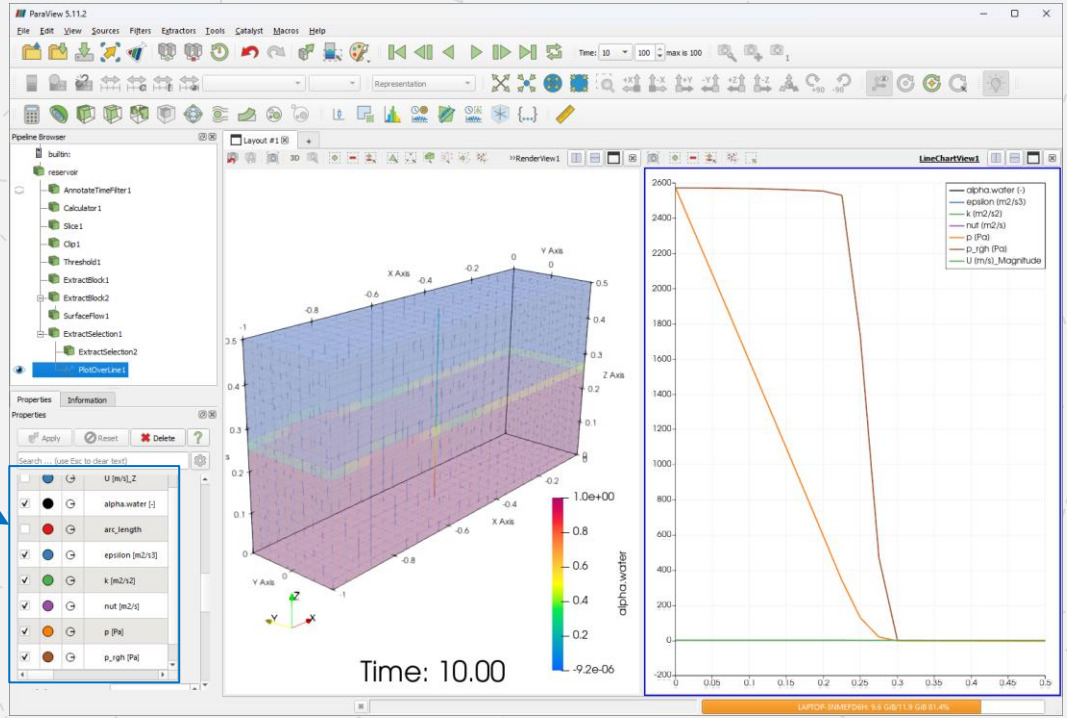


Plot over line

LineChartView will be displayed in a horizontal split canvas, with fields checked in the properties panel.

Y axis in the Line chart represent data values; X axis represent Line distance

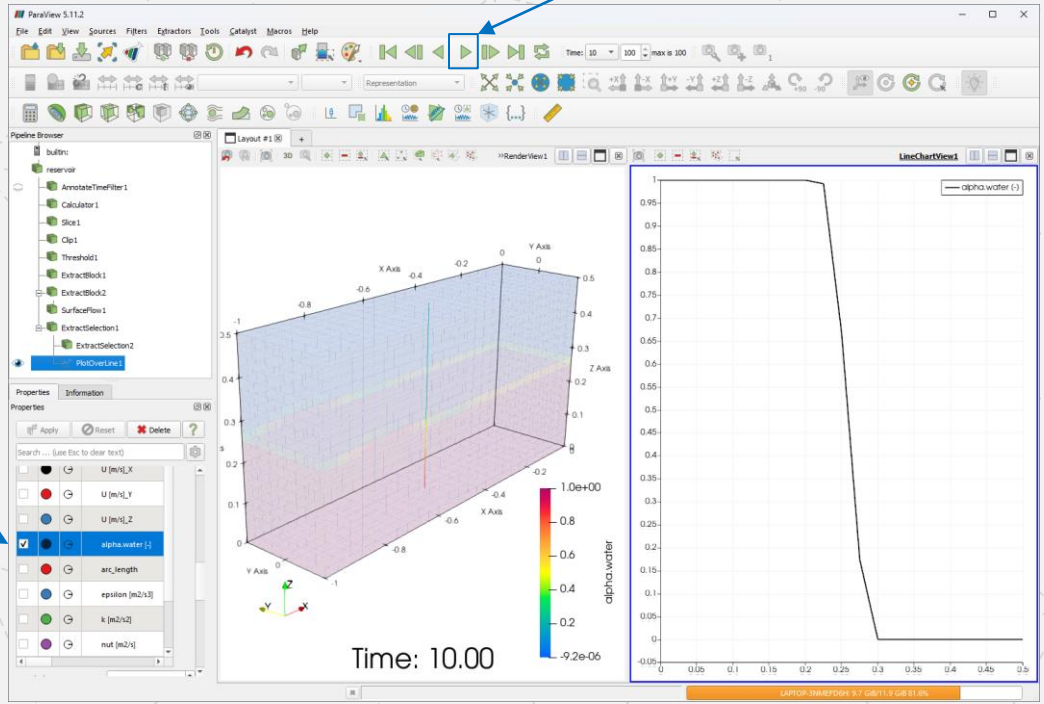
Analysed fields



Plot over line

Keep just “alpha.water [-] field checked and move forward in time to visualize chart evolution.

Alpha.water [-]



Probe location

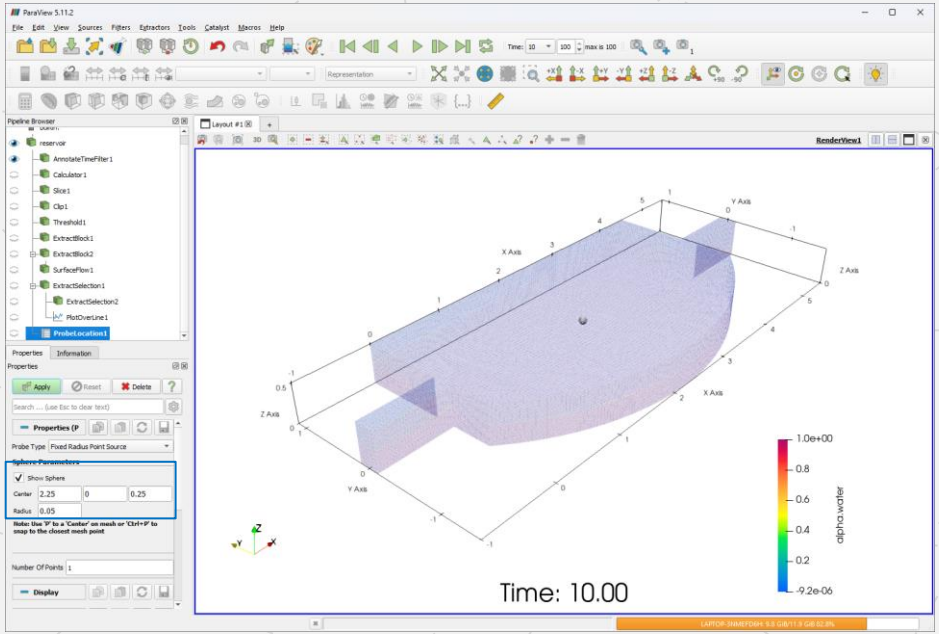
Select the Reservoir and apply the *Probe Location* filter.

Enter center point (2.25 0 0.5) in the properties panel. Change reservoir opacity to view the point location.

Check the “Axes grid” box in the properties panel of the Reservoir to view dimensions.

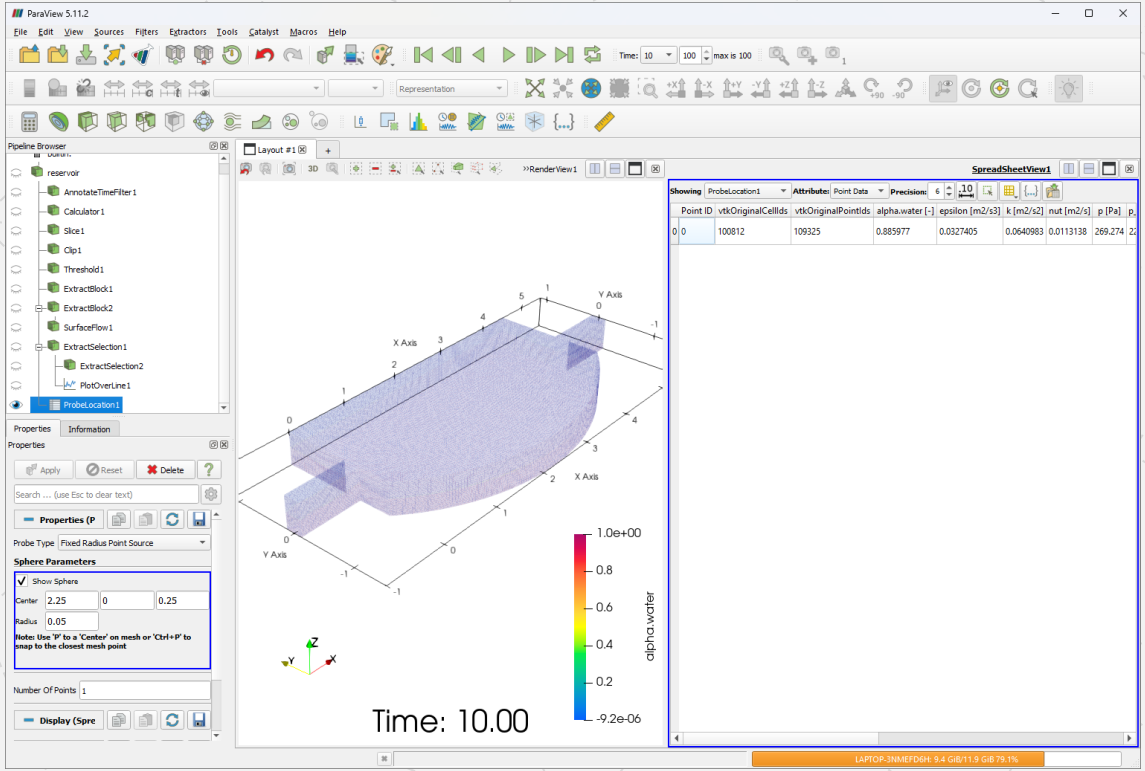
Center (2.25 0 0.25)

Radius 0.05



Probe location

SpreadSheetView will be displayed, with location information for the current instante of time.

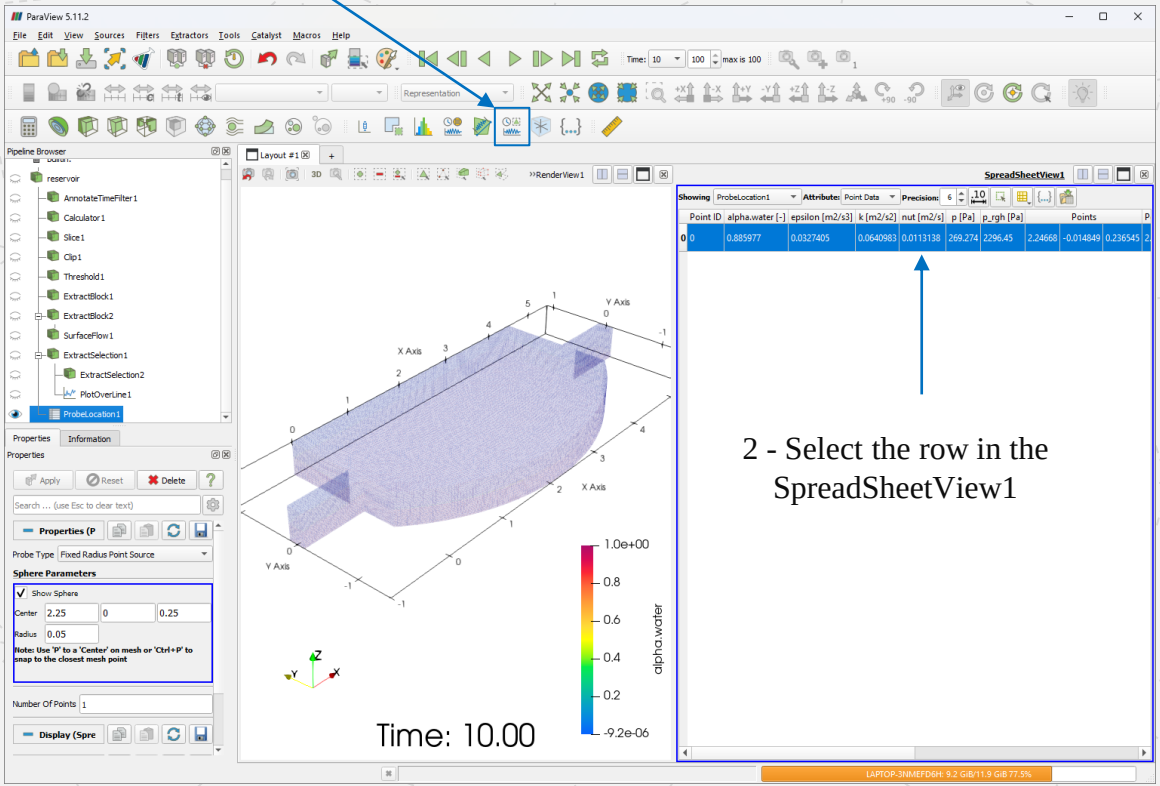


CRheo@IPC

Plot selection over time

3 - Select *Plot Selection Over Time* filter and click 

1 - Select ProbeLocation1

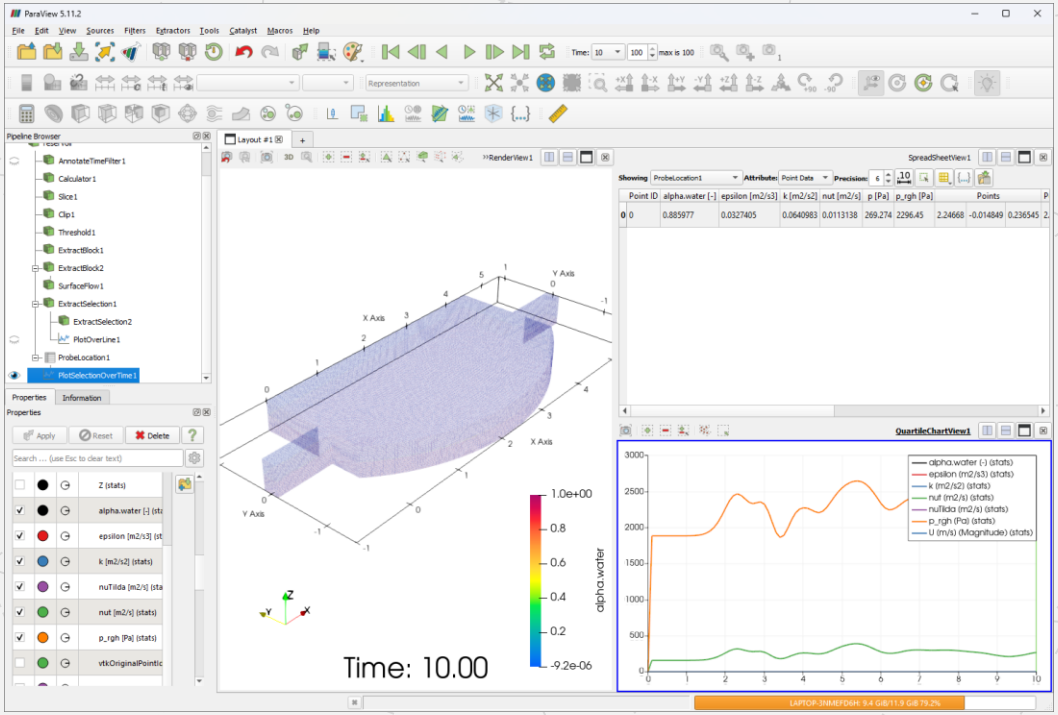


2 - Select the row in the SpreadSheetView1

Plot selection over time

QuartileChartView1 will be displayed in the canvas, with fields checked in the properties panel.

Y axis in the chart represent data values; X axis represent time.

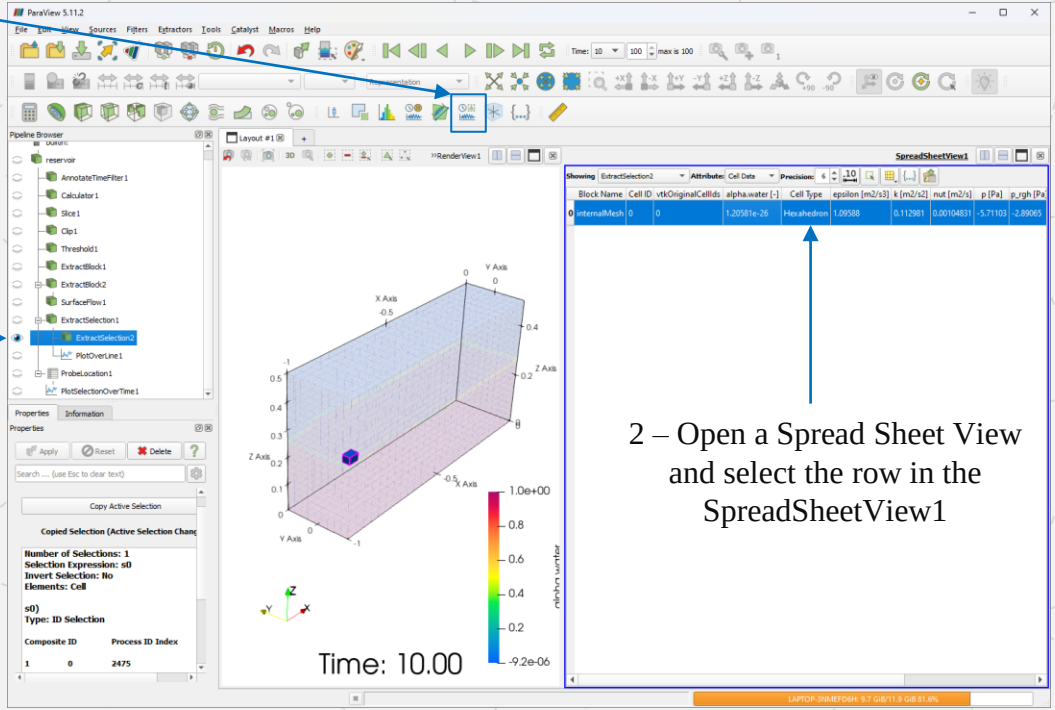


Plot selection over time

The same procedure can be done for a cell instead of a point. For that, open a Spread Sheet View of ExtractSelection2.

3 - Select *Plot Selection Over Time* filter and click 

1 - Select ExtractSelection2

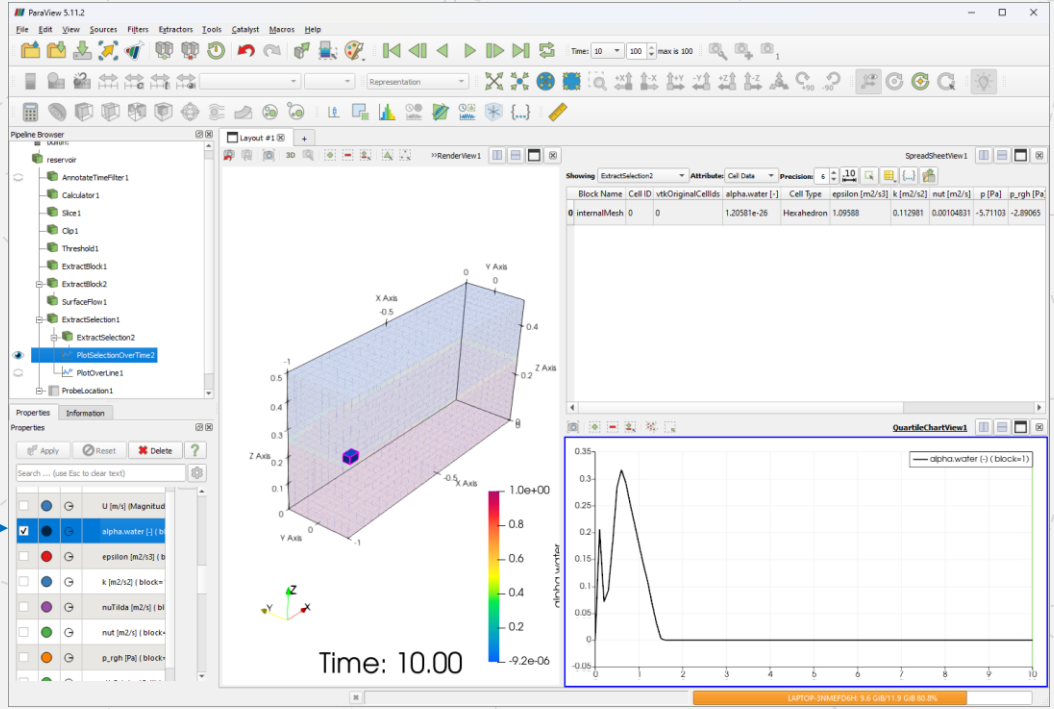


Plot selection over time

QuartileChartView1 will be displayed in the canvas, with fields checked in the properties panel.

Y axis in the chart represent data values; X axis represent time.

Check just
alpha.water [m/s]



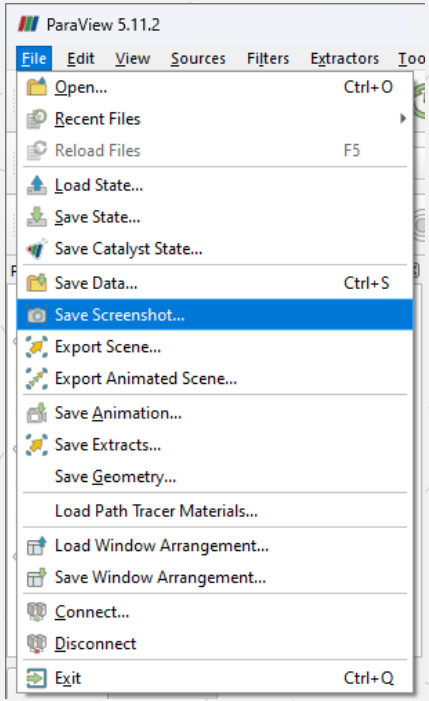
6

Saving progress

- Images
- Animations
- States

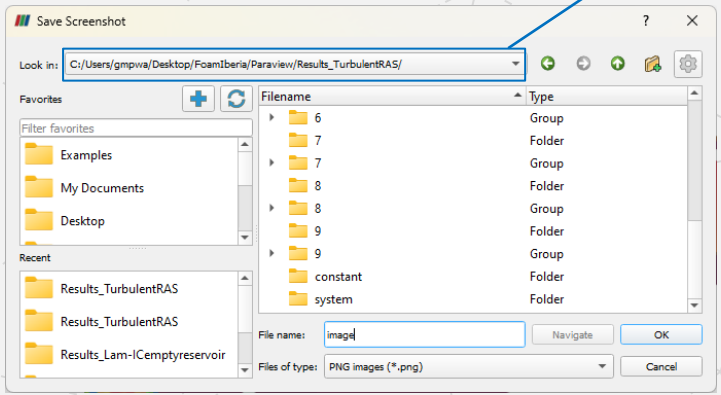
Images

(1)



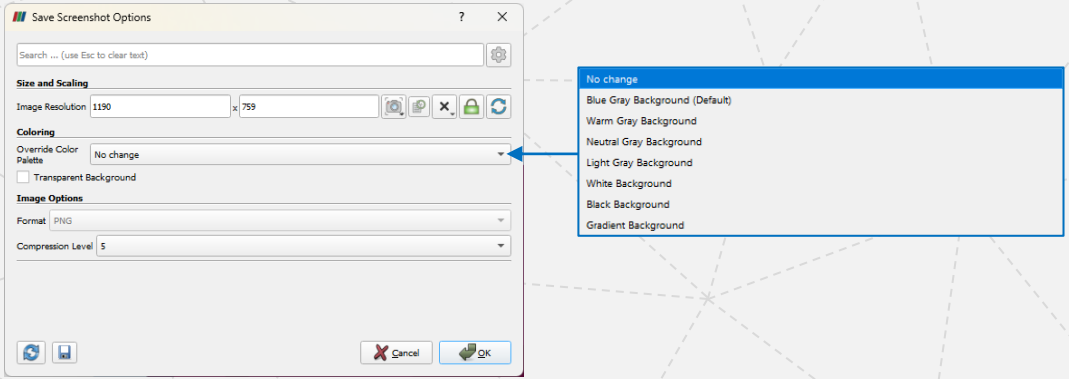
(2)

Saving location



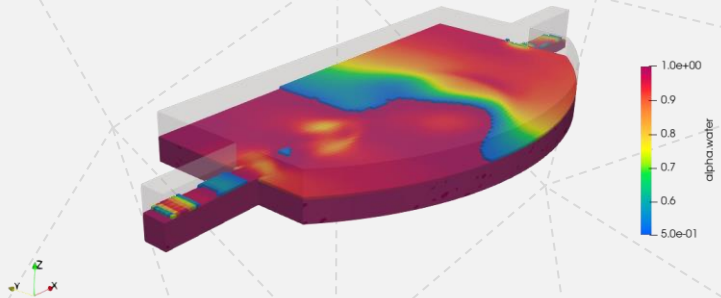
(3)

background

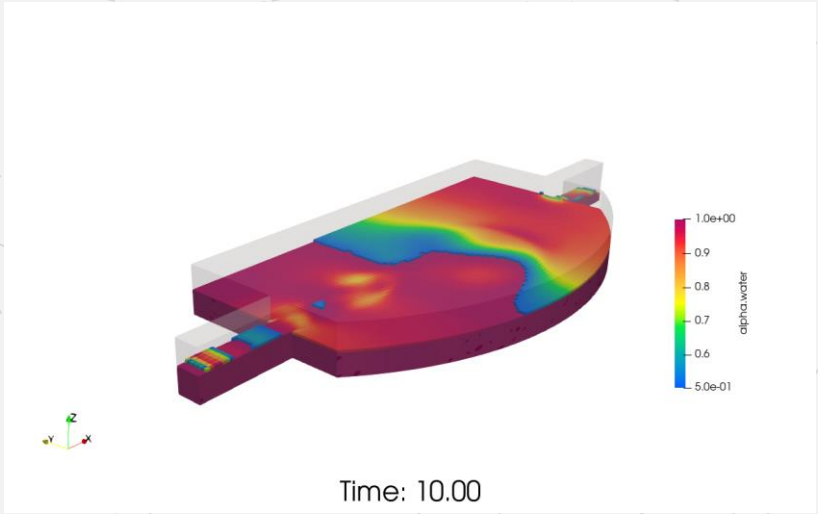


Images

Checked “Transparent background”

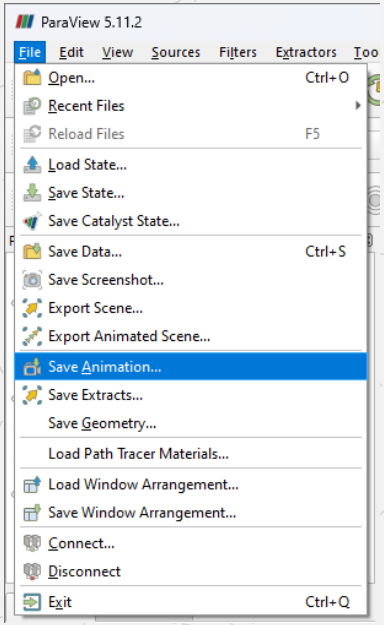


Unchecked “Transparent background”



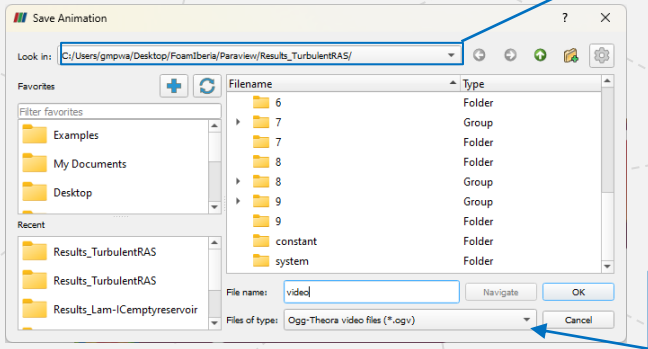
Animations

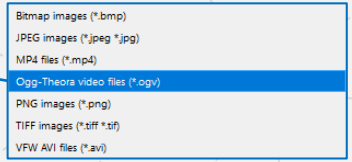
(1)



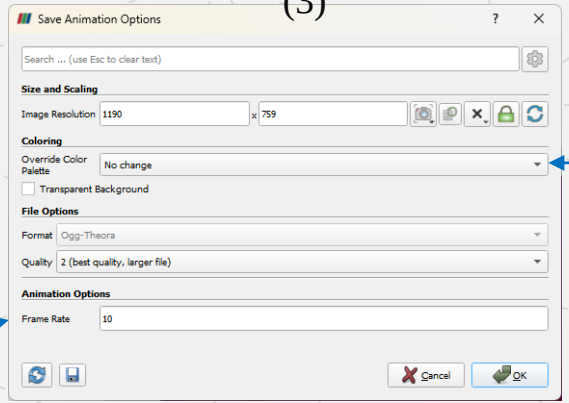
(2)

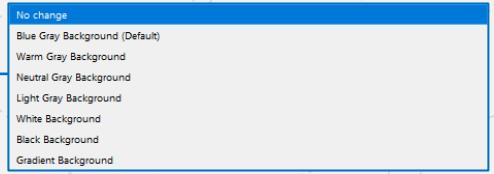
Saving location





(3)

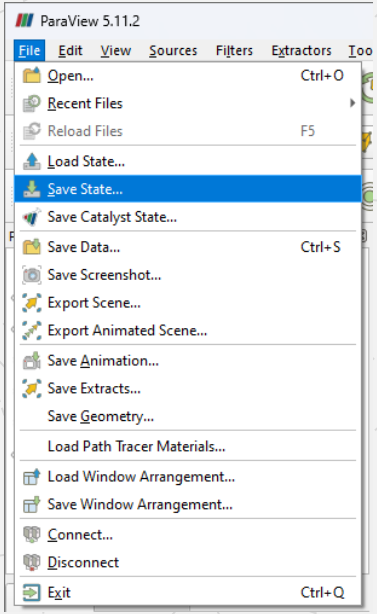




10 frames per second as timeStep is 0.1

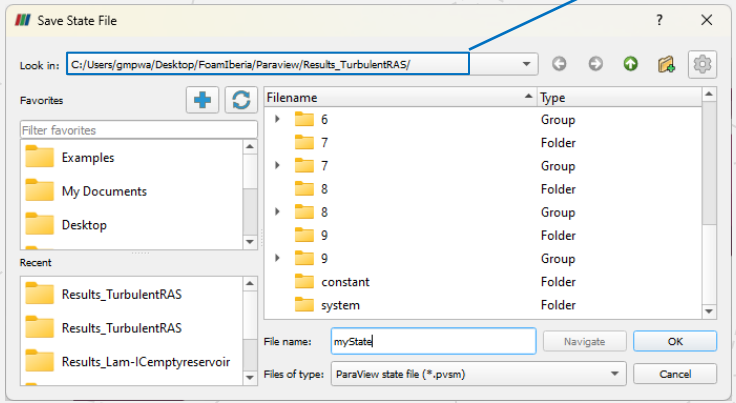
States

(1)



(2)

Saving location

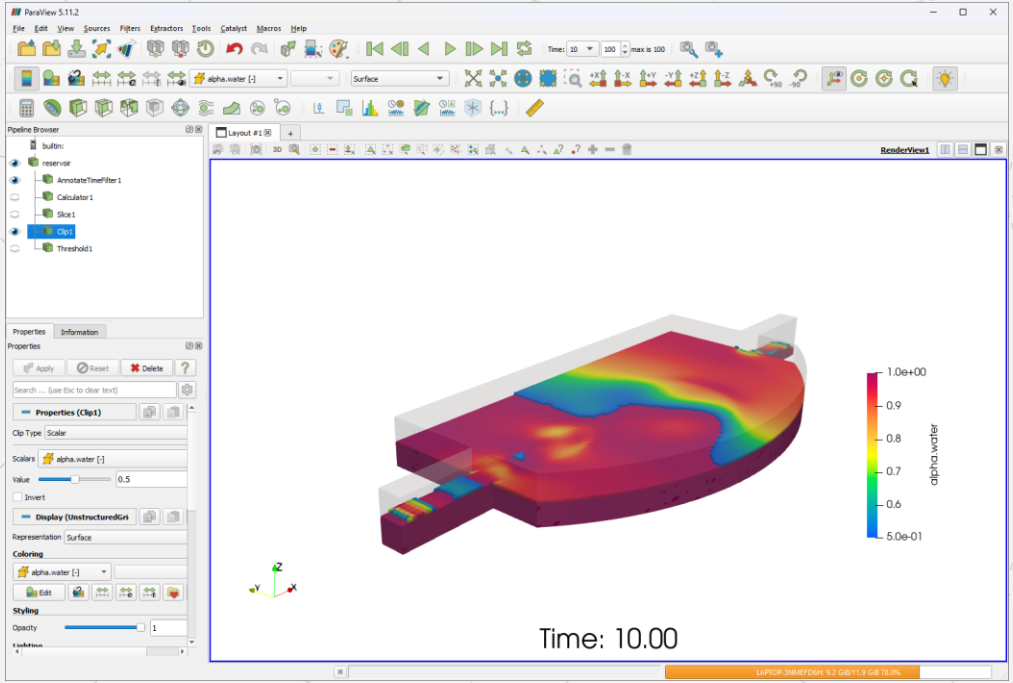


Saving a state allow you to reproduce your visualization settings and view without applying post-processing steps again.

It will save all your progress and it will work as a macro: once you load it, you will open your Paraview visualization the same way you saved it.

States

```
bino@LAPTOP-3NMEFD6H: /mnt/c/Users/gmpwa/Desktop/FoamIberia/Paraview/Results_TurbulentRAS
bino@LAPTOP-3NMEFD6H:/mnt/c/Users/gmpwa/Desktop/FoamIberia/Paraview/Results_TurbulentRAS$ Paraview myState.pvsm
```



States

The state file (.pvsm format) can be standardized to be fitted for other case studies, allowing to open a new case with the same applyed filters.

In the terminal, open the .pvsm file

> nano myState.pvsm

Press ctrl+W and search for “TurbulentRAS” (it will lead you to the line where it is written)

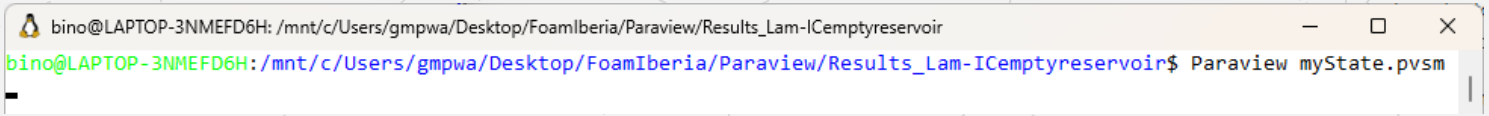
Replace the file location inside the “ ” for “.\open.foam”

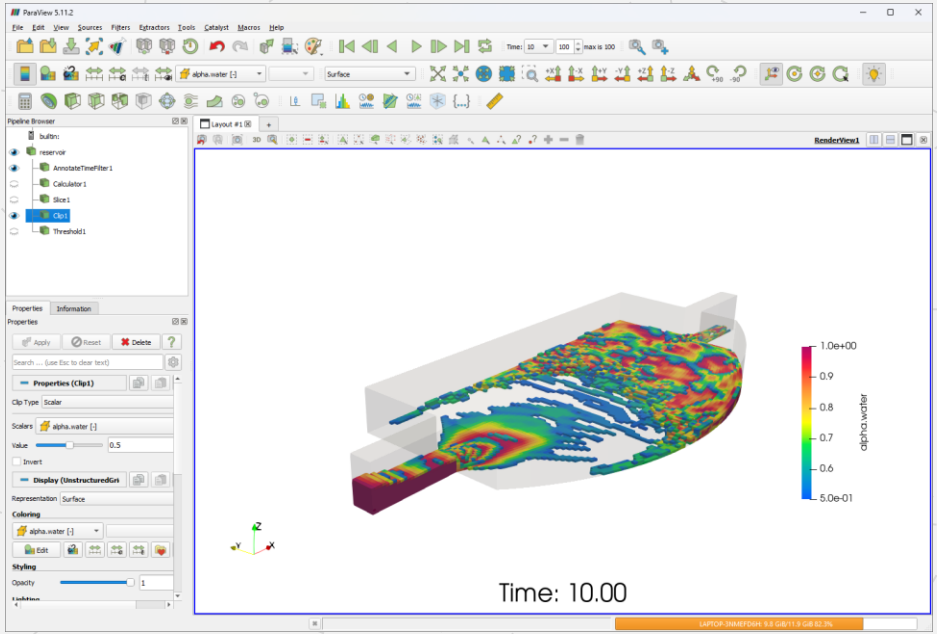
Press ctrl+X and then “Y” to exit and save

States

Copy the myState.pvsm file to the **Results_Lam_Icemptyreservoir** case folder and open it in Paraview

Note: you need to have an “open.foam” file in the **Results_Lam_Icemptyreservoir** folder.



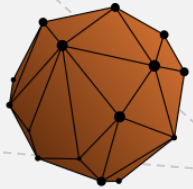


FOAM@Iberia 2023 5th edition

THANK YOU

This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software and owner of the OPENFOAM® and OpenCFD® trademarks.

Gabriel Mar Pinto Wagner
gmpwagner@gmail.com
p42786@alunos.uminho.pt



Computational
Rheology
@IPC

University of Minho
School of Engineering

iPC
INSTITUTE FOR
POLYMERS AND COMPOSITES