

LaMEM short course

17-21 02 2025 Heidelberg

Nicolas Riel- nriel@uni-mainz.de

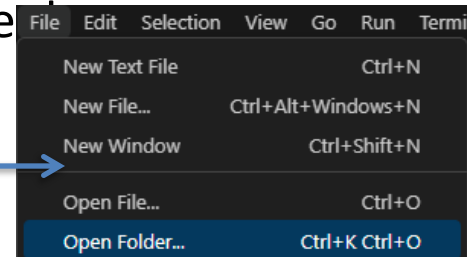
First LaMEM model

- Using VScode and the Julia REPL

→ go to 00_falling_block_3D/ directory using `cd ("path")` or use shell ; for Unix based systems

- you can also open folder in VScode

(this will place the REPL in the right location)



→ load LaMEM, provide input file and run test model

```
julia> using LaMEM

julia> paramFile=("FallingBlock_Multigrid.dat")
"FallingBlock_Multigrid.dat"

julia> run_lamem(paramFile,1)
```

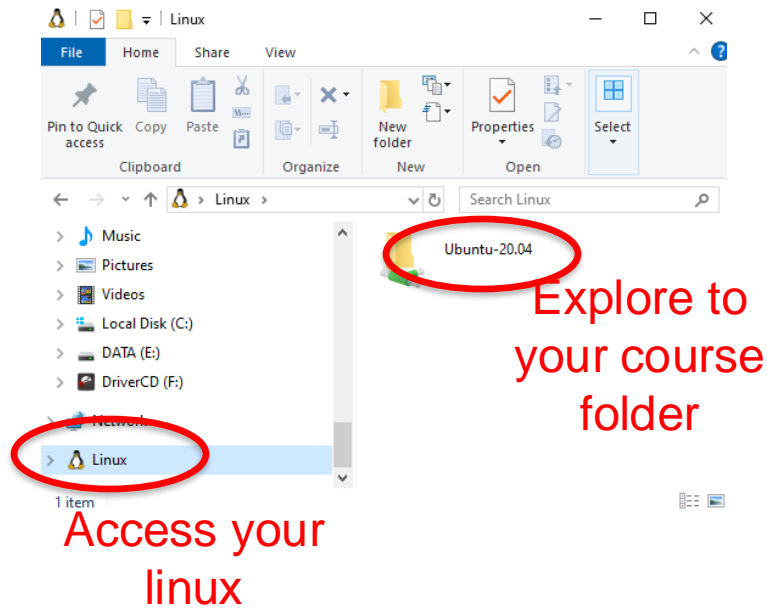
A few moments later...



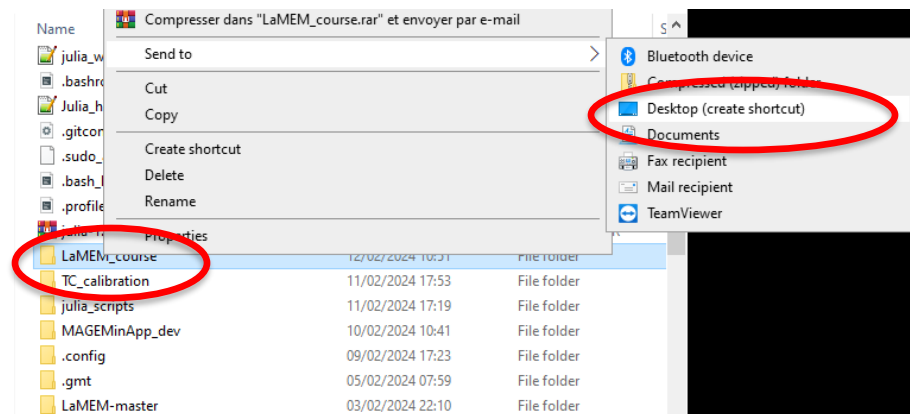
```
SMES solution time : 5.2892 (sec)
-----
Residual summary:
Continuity:
  |Div|_inf = 4.955148275832e-07
  |Div|_2   = 4.156760835818e-06
Momentum:
  |mRes|_2  = 1.842965364402e-03
-----
Actual time step : 9.70017 [ ]
-----
Marker control [0]: (AVD YZED) injected 8 markers and deleted 0 markers in 8.4110e-04 s
Marker control [0]: (AVD XZED) injected 12 markers and deleted 0 markers in 1.3592e-03 s
Marker control [0]: (AVD XYED) injected 11 markers and deleted 0 markers in 1.4619e-03 s
-----
Saving output ... done (1.05814 sec)
-----
===== SOLUTION IS DONE! =====
-----
Total solution time : 53.4271 (sec)
-----
julia> |
```

Visualize your first simulation using Paraview

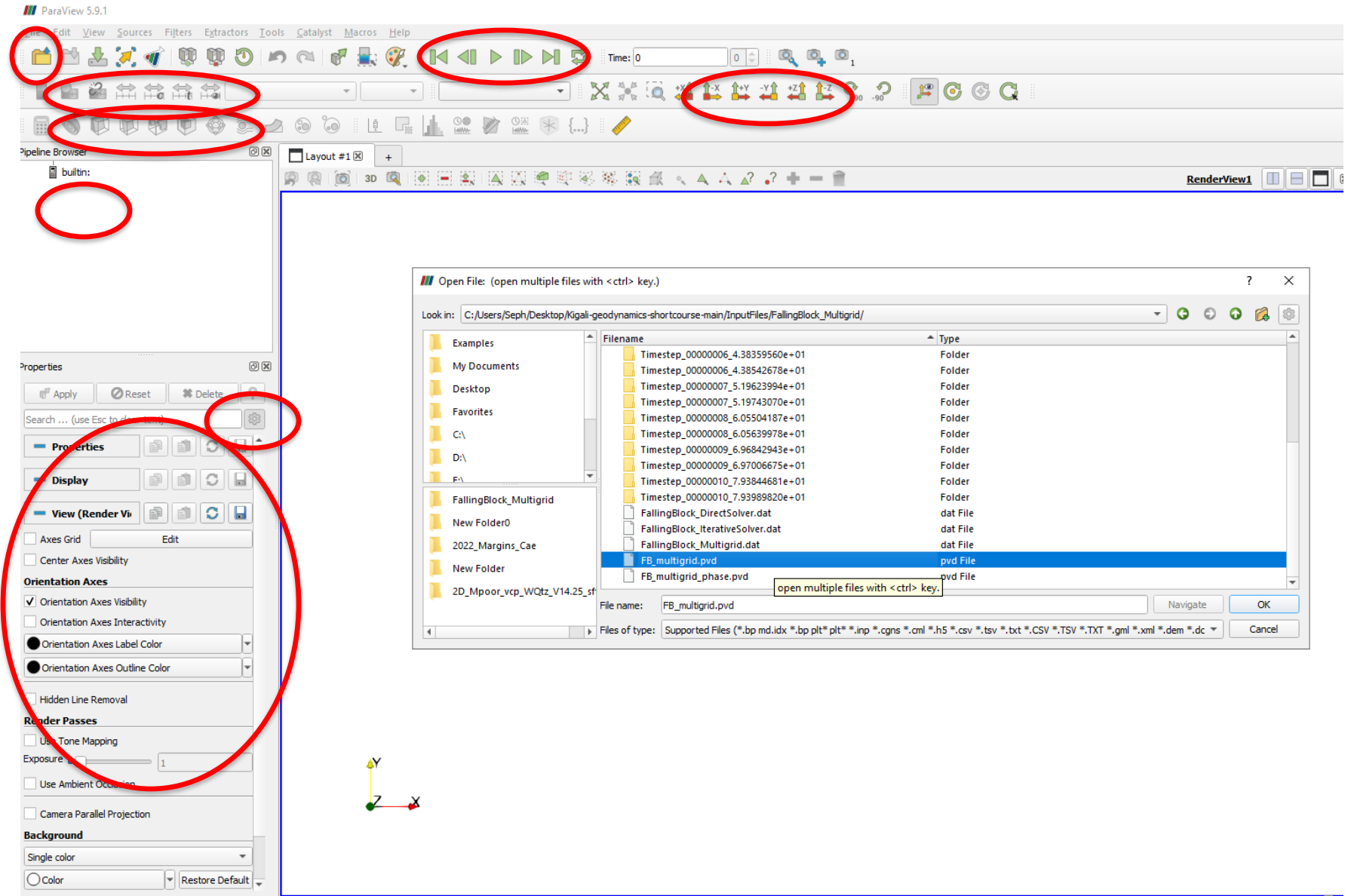
How to access the folder using Paraview? (Windows)



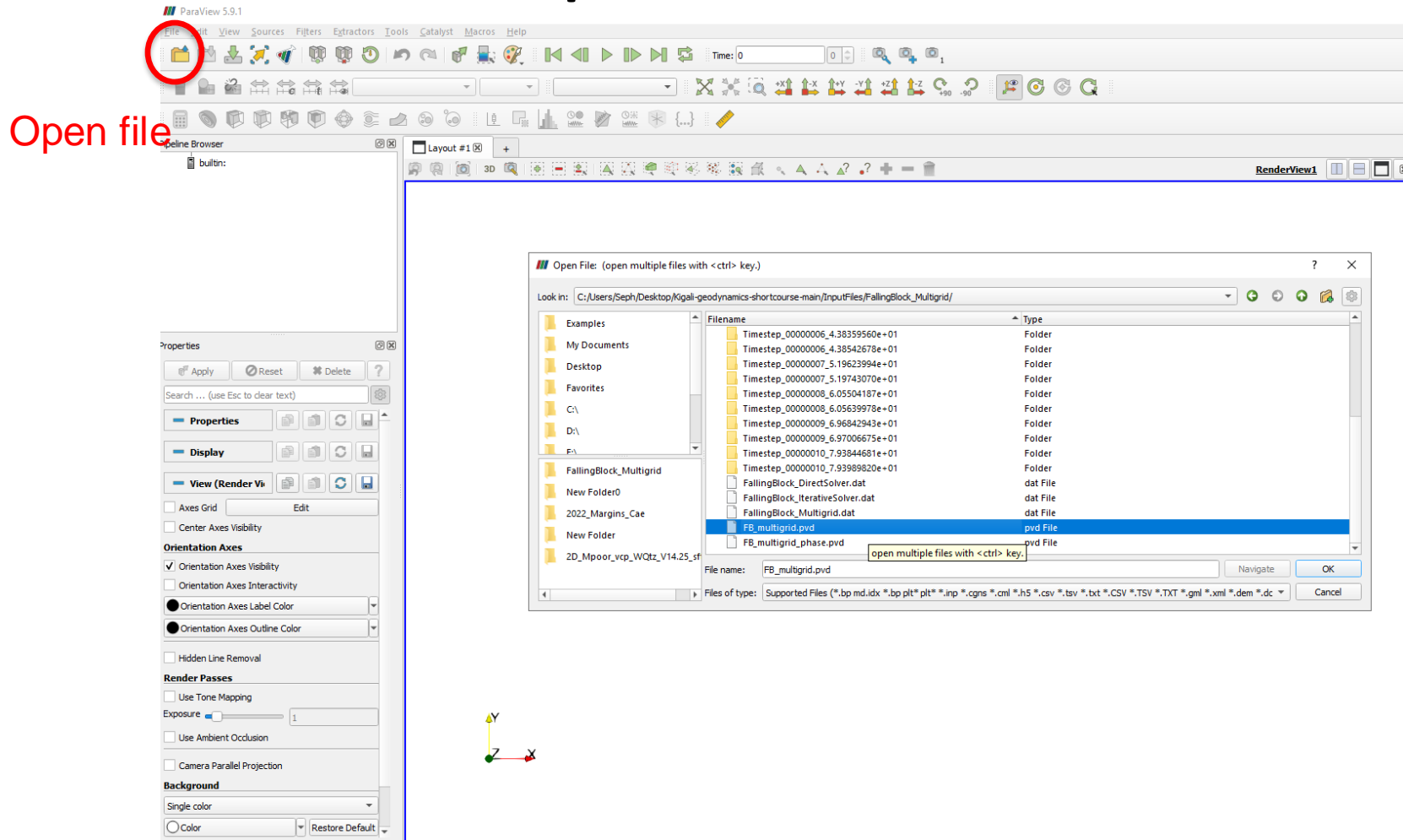
Right click
and send a
short cut to
Desktop



Paraview: overview



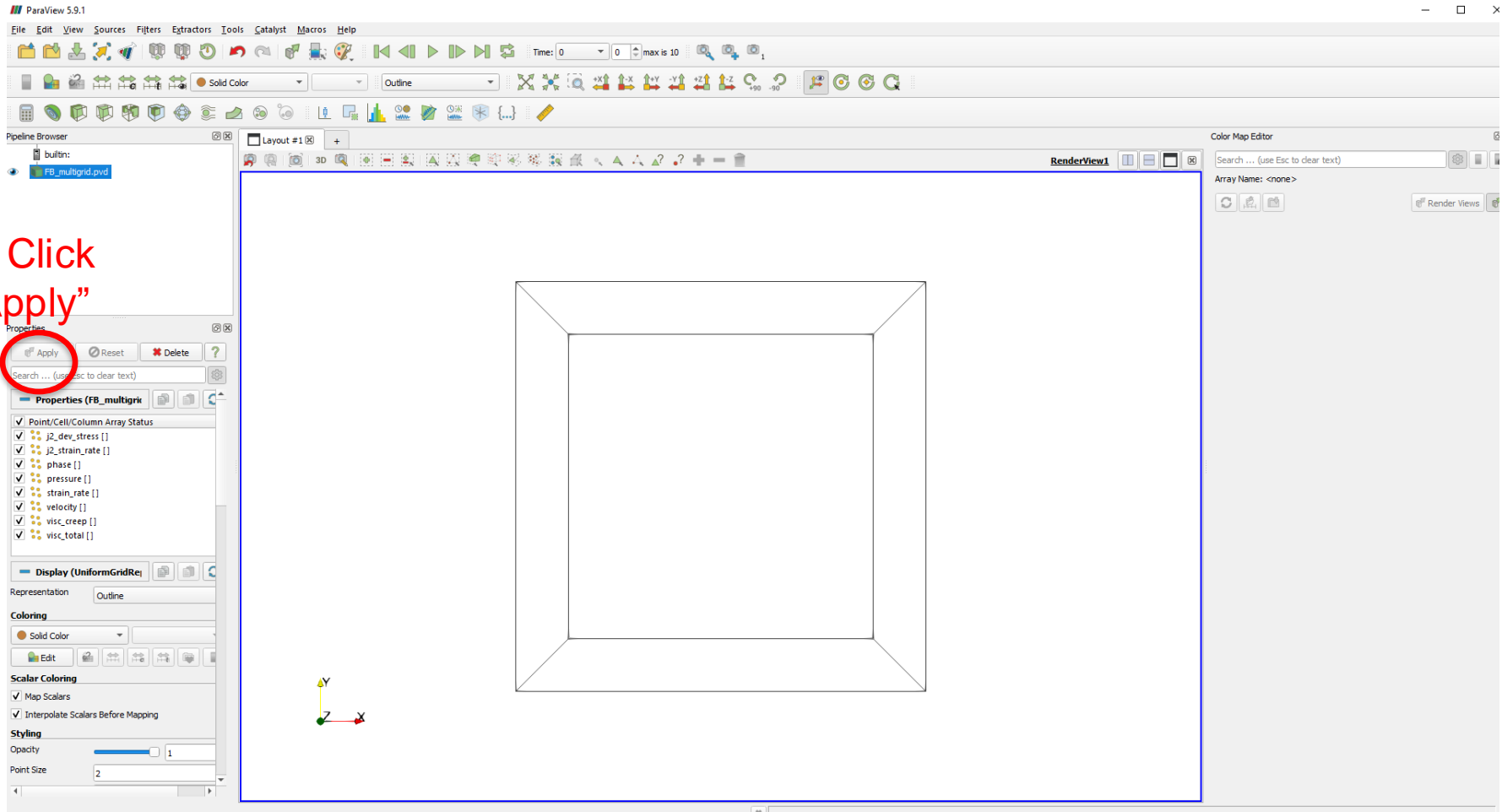
Open Paraview



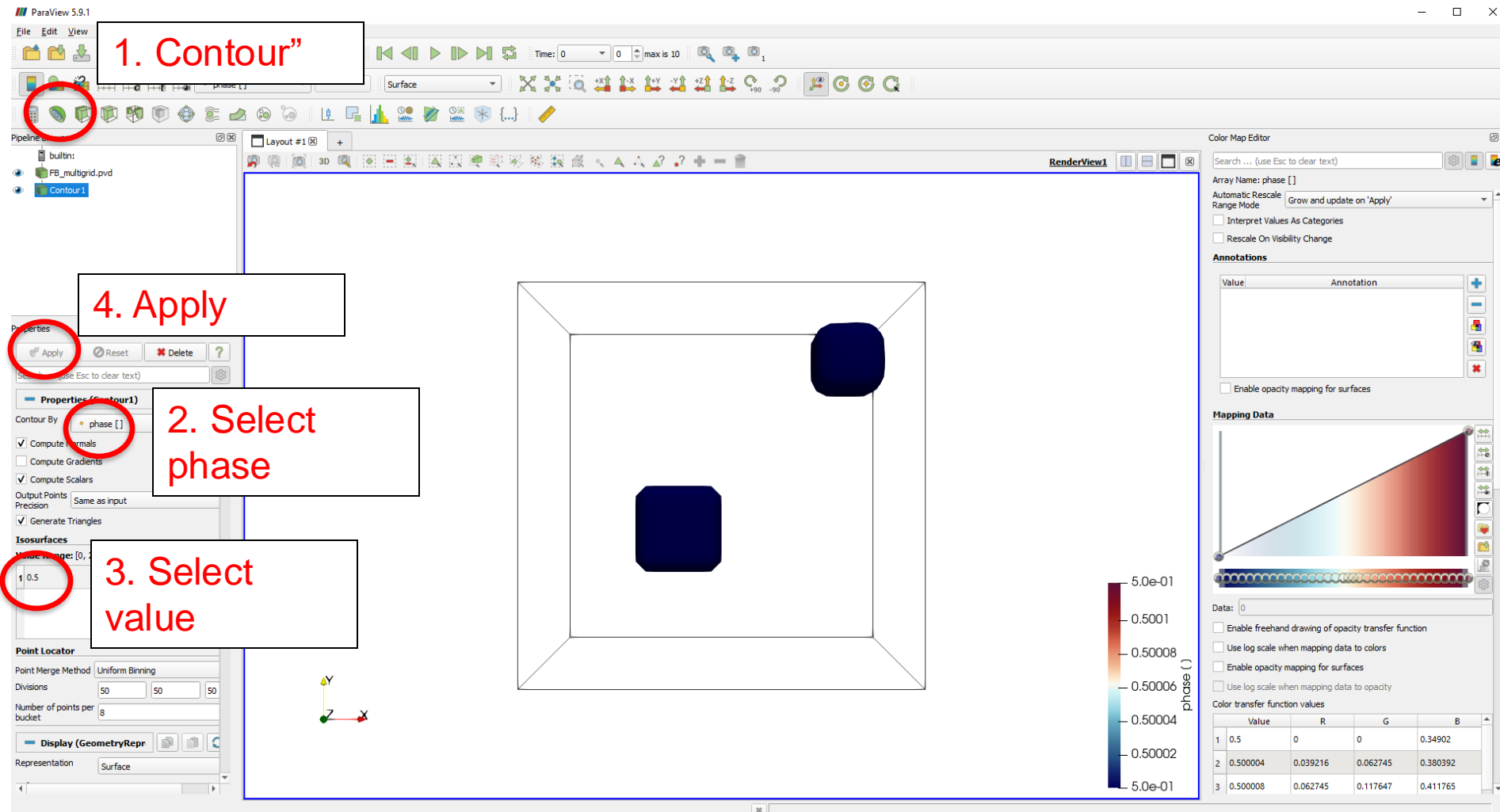
- Open the file “FB_multigrid.pvd” in the folder you just downloaded

Load data

1. Click
“Apply”



Display falling blocks



Change block color

The screenshot displays the ParaView 5.9.1 software interface. The top toolbar features a 'Solid Color' button, which is circled in red. A red arrow points from this button to a text box stating: '1. While Contour1 is selected change this field to Solid Color'. In the 'Pipeline Browser' on the left, 'Contour1' is selected and highlighted in blue. Another red arrow points from this selection to a text box stating: '2. Activate advanced settings'. Below this, in the 'Properties' panel, the 'Advanced' settings icon (a gear) is circled in red. A third red arrow points from this icon to a text box stating: '3. Edit color'. In the 'Properties' panel, under the 'Coloring' section, the 'Edit' button is circled in red. The main 3D view on the right shows two yellow rectangular blocks within a wireframe cube, representing the result of the color change.

1. While Contour1 is selected change this field to Solid Color

2. Activate advanced settings

3. Edit color

- Note that any field can be contoured: including phase, temperature etc.

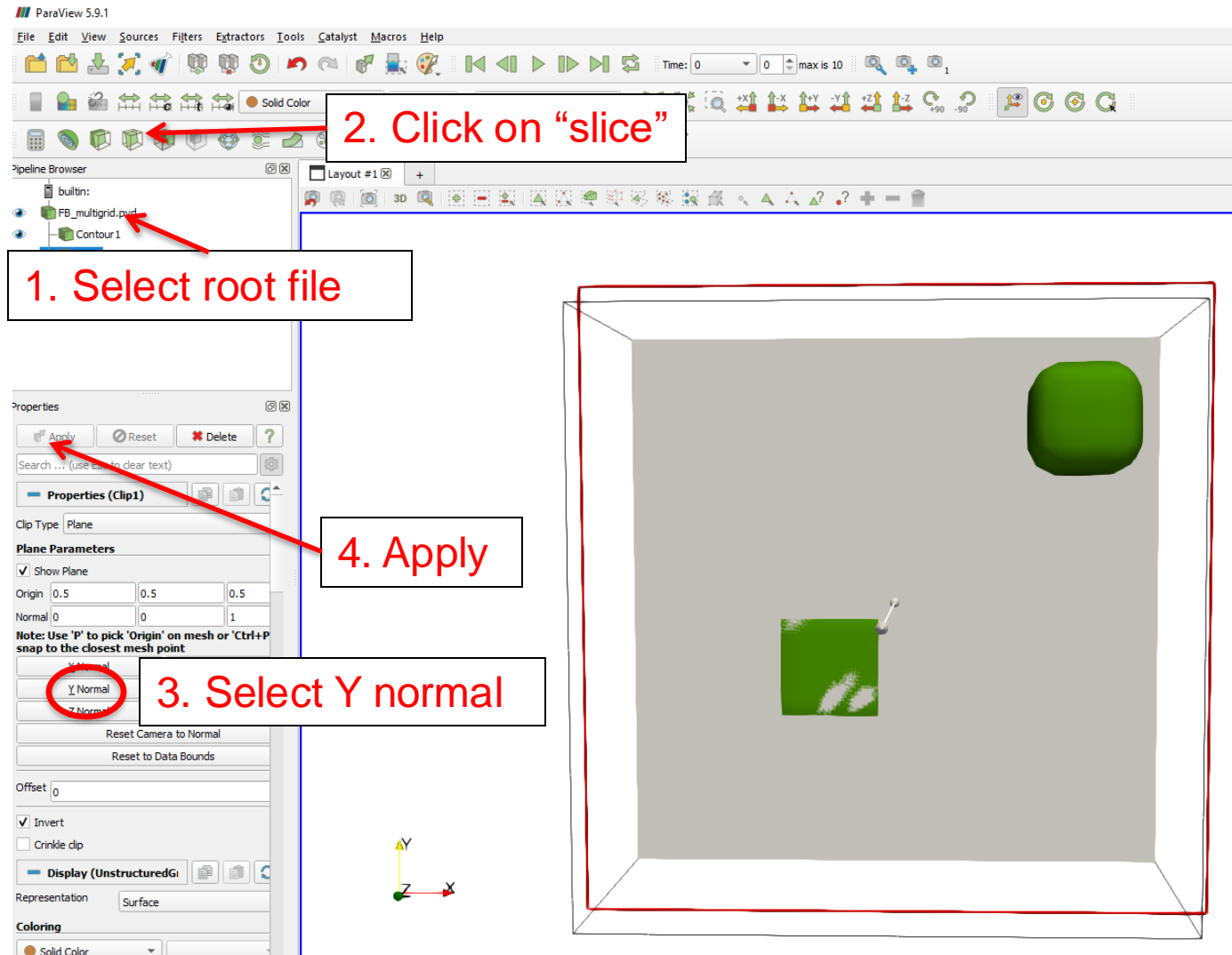
Clip host material

The image shows the ParaView 5.9.1 interface with four numbered steps for clipping a host material:

- 1. Select root file**: A red arrow points to the 'Clip1' entry in the Pipeline Browser.
- 2. Click on "clip"**: A red arrow points to the 'clip' button in the top toolbar.
- 3. Select Y normal**: A red circle highlights the 'Y Normal' option in the 'Plane Parameters' section of the Properties panel.
- 4. Apply**: A red arrow points to the 'Apply' button in the Properties panel.

The main 3D view displays a green cube and a green rectangular plane within a wireframe box. A small 3D coordinate system (X, Y, Z) is visible at the bottom left of the 3D view.

Or slice material



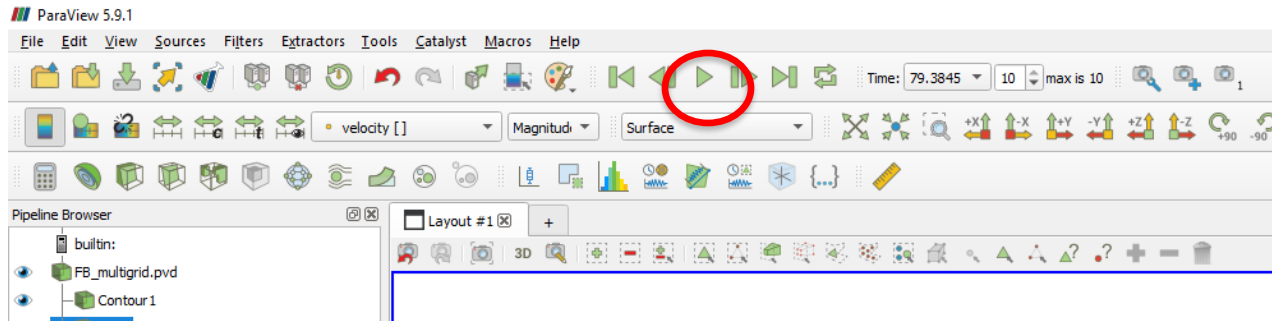
Display field

The image shows the ParaView 5.9.1 interface with five numbered instructions for displaying a velocity field:

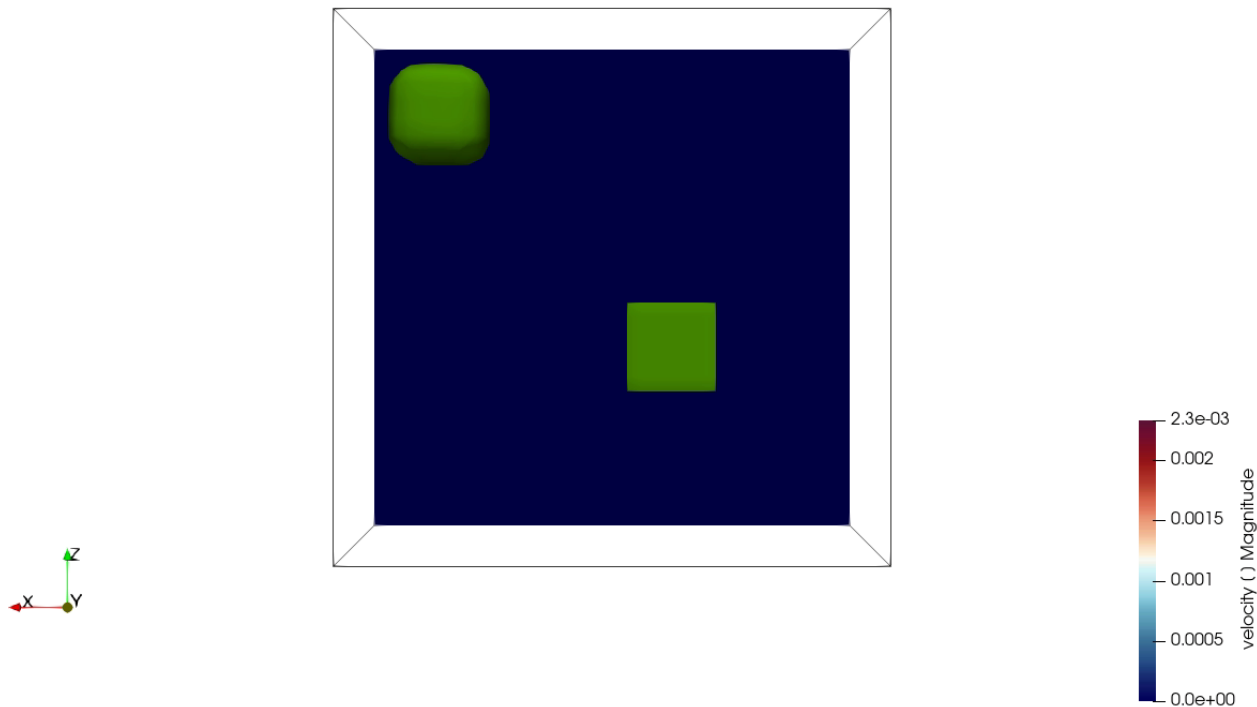
- 1. Make sure "Surface" is selected**: Points to the "Surface" dropdown menu in the top toolbar.
- 2. Select velocity**: Points to the "velocity []" dropdown menu in the top toolbar.
- 3. Untick show plane**: Points to the "Show Plane" checkbox in the Properties panel for "Clip1".
- 4. Click rescale over all timestep -> rescale**: Points to the "rescale" button in the top toolbar.
- 5. Set view direction to Y**: Points to the "Y" view direction button in the top toolbar.

The main 3D view displays a velocity field visualization with a green cube and a green plane. The Properties panel shows the "Clip1" properties, including "Show Plane" (unchecked), "Origin" (0.5, 0.5, 0.5), and "Normal" (0, 1, 0).

Play the timesteps



... it should look like this:



Change colormap

2. Choose a colormap

1. Open the colormap preset

3. Apply

4. Close

The screenshot shows the ParaView software interface. The 'Choose Preset' dialog box is open, displaying a list of color maps. The 'Mapping Data' section on the right is also visible, showing a color bar for 'velocity () Magnitude' ranging from 0.0e+00 to 3e-03. The 'Color transfer function values' table is shown at the bottom right.

	Value	R	G	B
1	0	0.101961	0.101961	0.101961
2	0.000145839	0.227451	0.227451	0.227451
3	0.000291678	0.359939	0.359939	0.359939

Add velocity vectors (Glyphs) part 1

The screenshot shows the ParaView 5.9.1 interface with the following components and annotations:

- 2. Click on glyphs:** A red arrow points to the 'Glyphs' icon in the top toolbar.
- 1. Select root file (contains the whole volume):** A red arrow points to 'FB_multigrid.pvd' in the Pipeline Browser.
- 5. Apply:** A red arrow points to the 'Apply' button in the Properties panel.
- 3. Select the orientation array to be velocity:** A red arrow points to the 'velocity []' option in the 'Orientation Array' dropdown.
- 4. Below in "masking" decrease the number of sample points to 3000:** A red arrow points to the 'Scale Factor' slider in the 'Masking' section.

The main 3D view displays a complex, swirling flow field represented by a dense collection of grey arrows (glyphs) originating from a green volume.

Add velocity vectors (Glyphs) part 2

ParaView 5.9.1

File Edit View Sources Filters Extractors Tools Catalyst Macros Help

Time: 79.3845 10 max is 10

velocity [] Magnitude Surface

Pipeline Browser

builtin:

- FB_multigrid.pvd
- Contour1
- Clip1
- glyph1

Layout #1

1. In the field select "velocity"

2. Then hide the clip

Properties

Apply Reset Delete

Search ... (use Esc to clear text)

Factor 0.1

Glyph Transform

Glyph Transform Transform2

Translate 0 0 0

Rotate 0 0 0

Scale 1 1 1

Masking

Glyph Mode Uniform Spatial Distribution (Bounds Base)

Maximum Number Of Sample Points 3000

Seed 10339

Display (GeometryRepre)

Representation Surface

Coloring

velocity.F1 Magnitude

1.6e-03

0.001

Magnitude

3. You can play the timesteps, change colormap ...

4. Similarly Glyphs can be applied to slices and clips

pg+00

Streamlines part 1

The screenshot displays the ParaView 5.9.1 software interface. The top toolbar contains various icons for file operations, viewing, and processing. The Pipeline Browser on the left shows a sequence of filters: 'built-in', 'FB_multigrid.pvd', 'Contour1', 'Clip1', 'Glyph1', and 'StreamTracer1'. The 'StreamTracer1' filter is highlighted in blue. A red arrow points from the 'StreamTracer1' filter in the Pipeline Browser to a text box labeled '1. Hide clip, slice and glyphs'. Another red arrow points from the 'StreamTracer1' filter to a text box labeled '2. Click on streamlines'. The Properties panel on the left shows the settings for the 'StreamTracer1' filter. The 'Seed Type' is set to 'Point Cloud'. A red arrow points from the 'Point Cloud' option to a text box labeled '3. Select see type: "Point cloud"'. The 3D view on the right shows a wireframe cube containing two green cubes and a wireframe sphere. A red arrow points from the 'Point Cloud' option in the Properties panel to the wireframe sphere in the 3D view.

ParaView 5.9.1

File Edit View Sources Filters Extractors Tools Catalyst Macros Help

Time: 79.3845 10 max is 10

Representation

Pipeline Browser

Layout #1

2. Click on streamlines

1. Hide clip, slice and glyphs

Properties

Apply Reset Delete

Search ... (use Esc to clear text)

Maximum Steps 2000

Maximum Streamline Length 1

Terminal Speed 1e-12

☒ Compute Vorticity

Seeds

Seed Type Point Cloud

Point Cloud Parameters

☒ Show Sphere

Center 0.5

Radius 0.1

Note: Use 'P' to a 'Center' on mesh or 'Ctrl+P' to snap to the closest mesh point

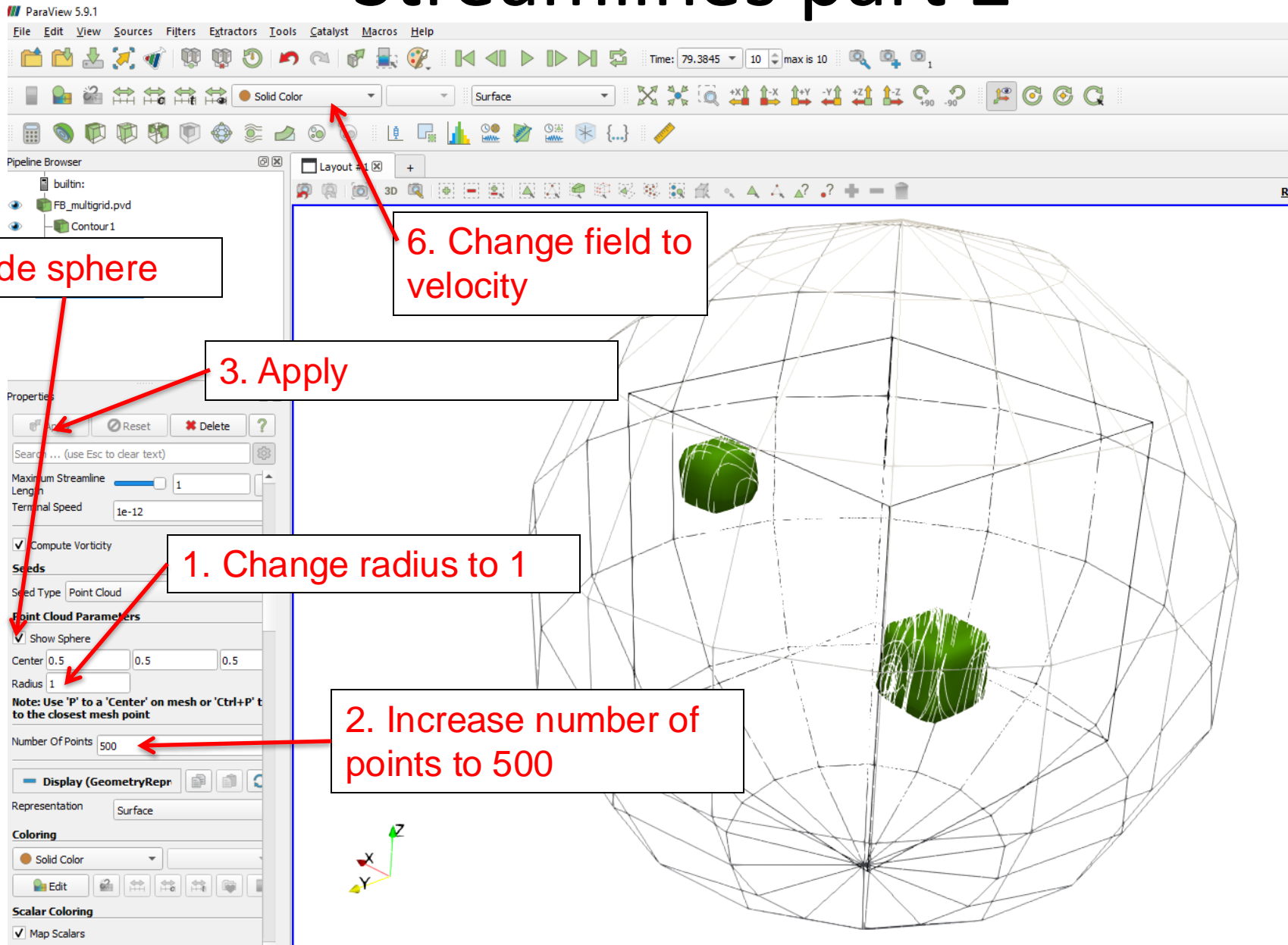
Number Of Points 100

Display

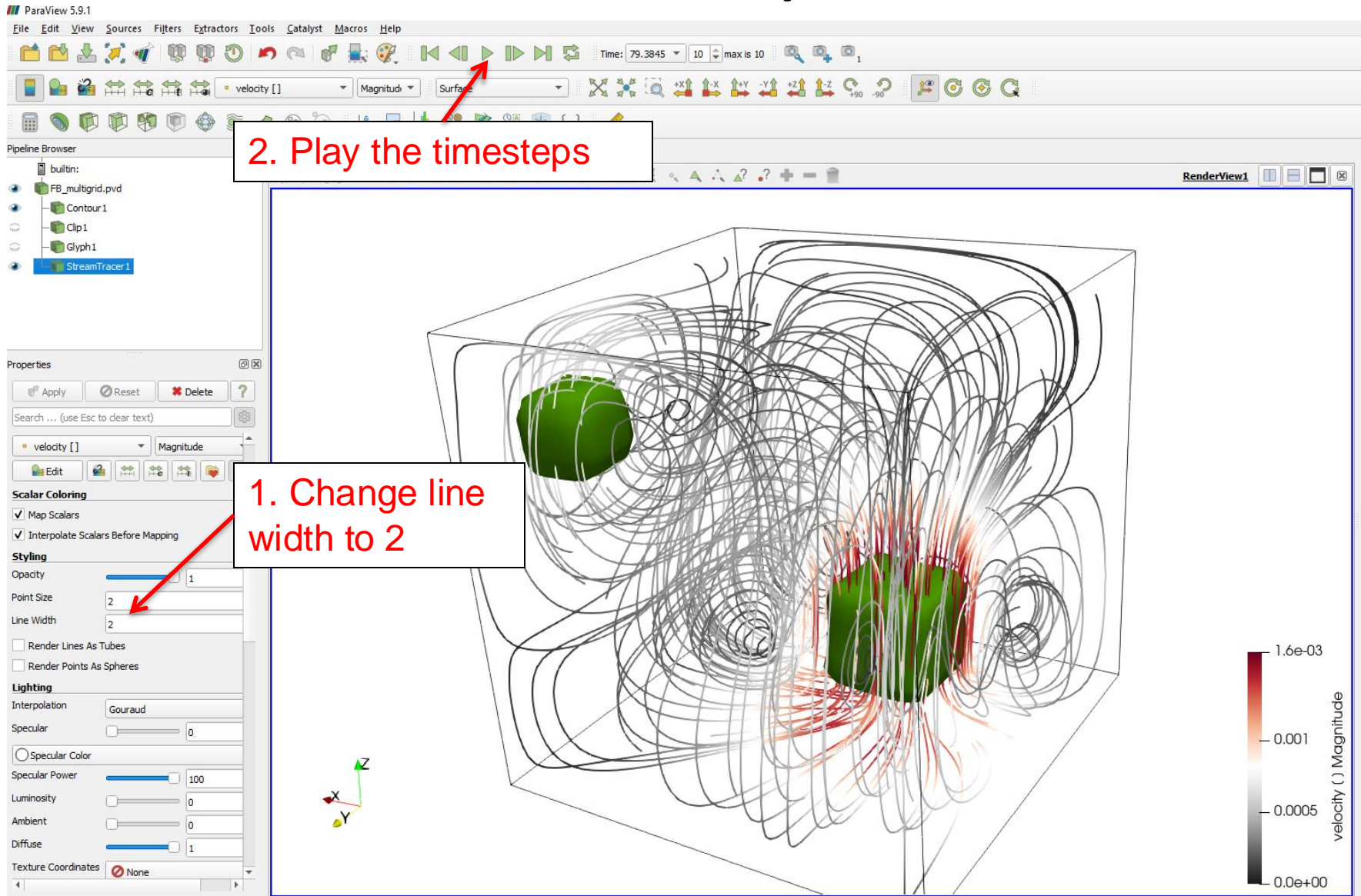
View (Render Vi

3. Select see type: "Point cloud"

Streamlines part 2

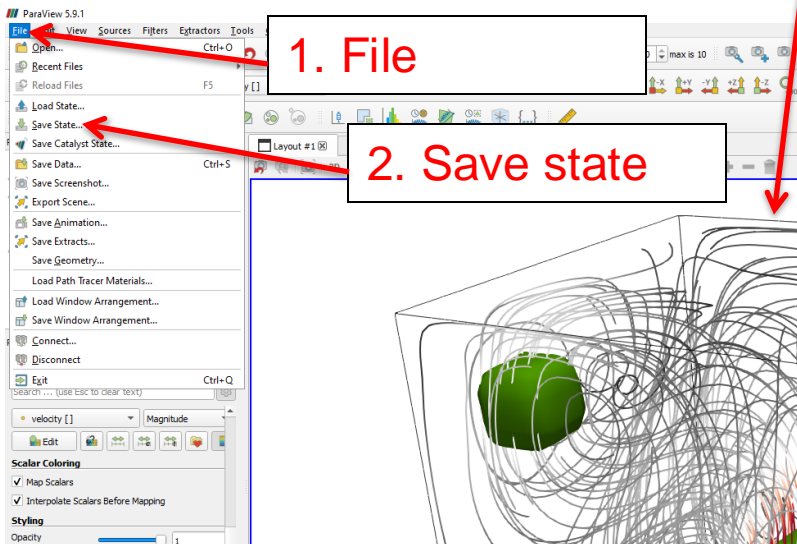
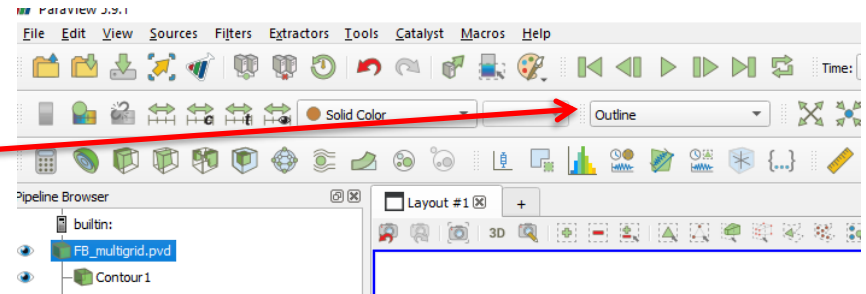


Streamlines part 3



Paraview useful tips

- Note that we did not change the root file display from “outline”. This allow to see the edges of the modeling volume
- If you are happy with the visualization output you can save the state (.pvsm file)



- Later you can reset/relaunch paraview then file → load state and select a model directory (it does not have to be the same model output!)

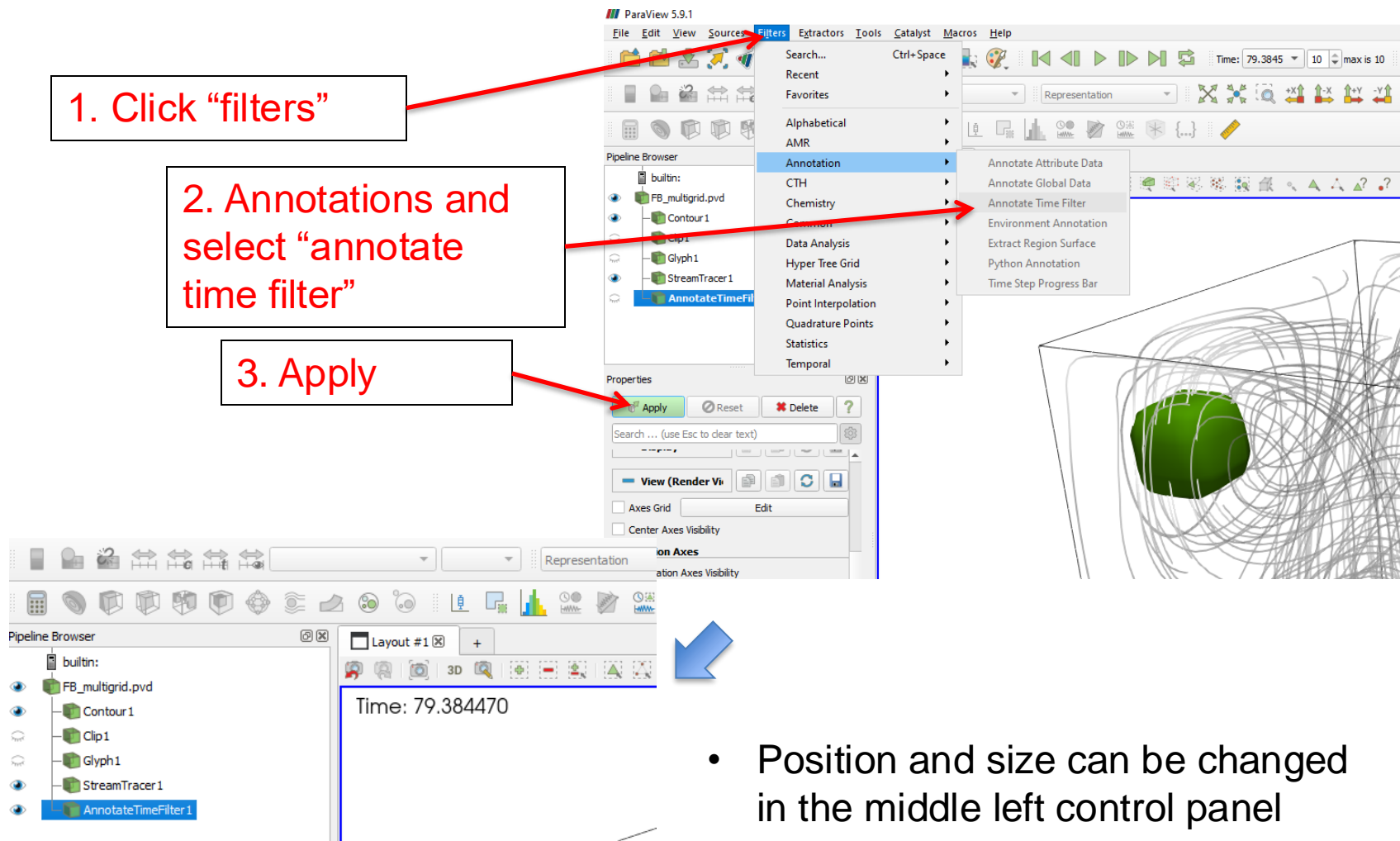
Paraview useful tips

- When producing figures with paraview it is often useful to display the modeling time, this can be done using a time filter:

1. Click “filters”

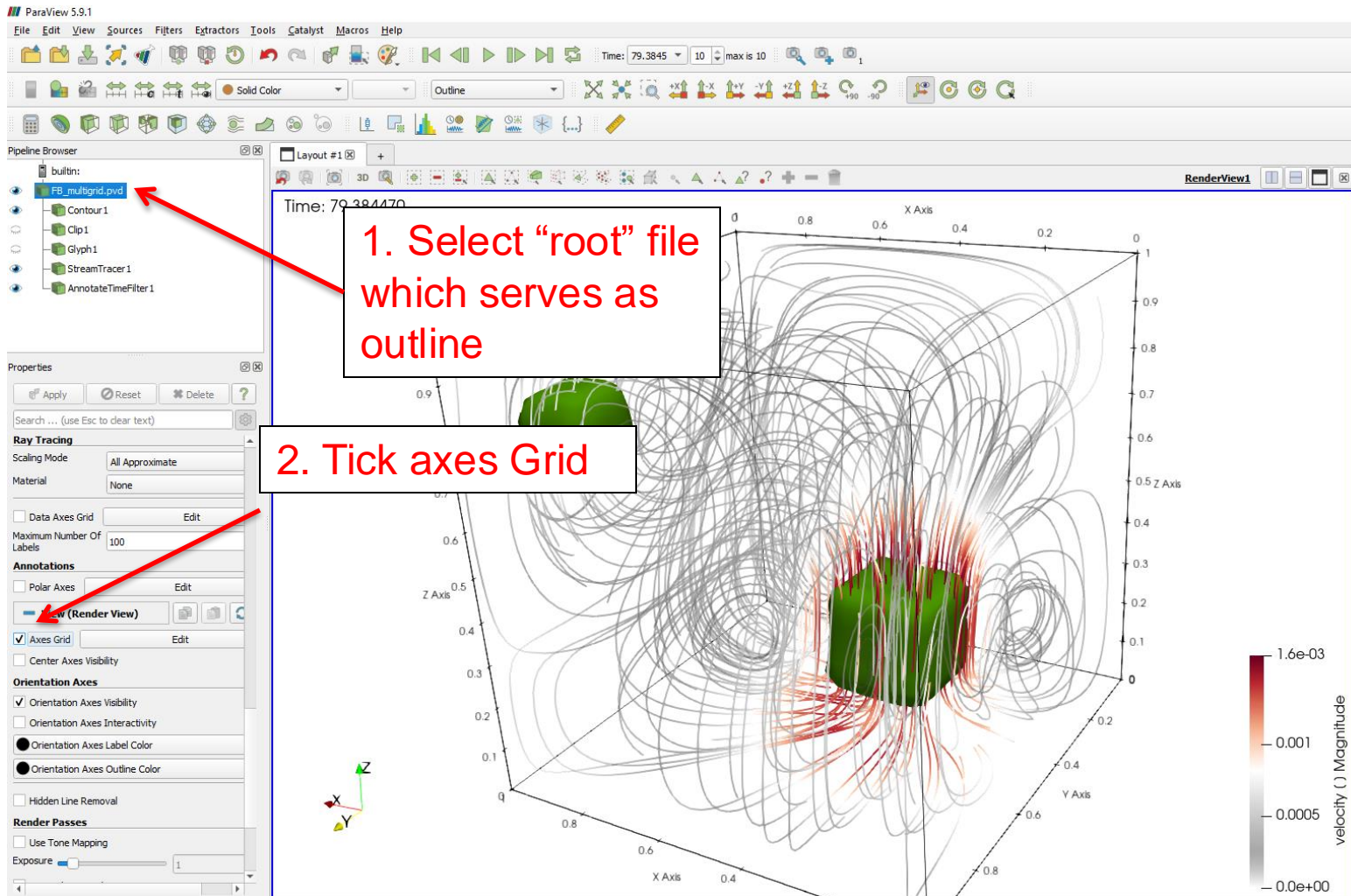
2. Annotations and select “annotate time filter”

3. Apply



Paraview useful tips

- Model dimensions can be useful too...



Paraview useful tips

- Screenshots and animation can be saved in the “File” tab

