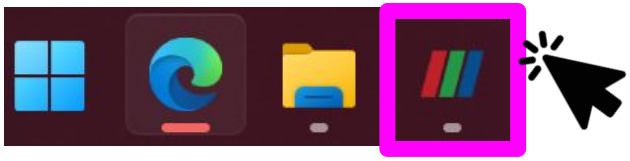
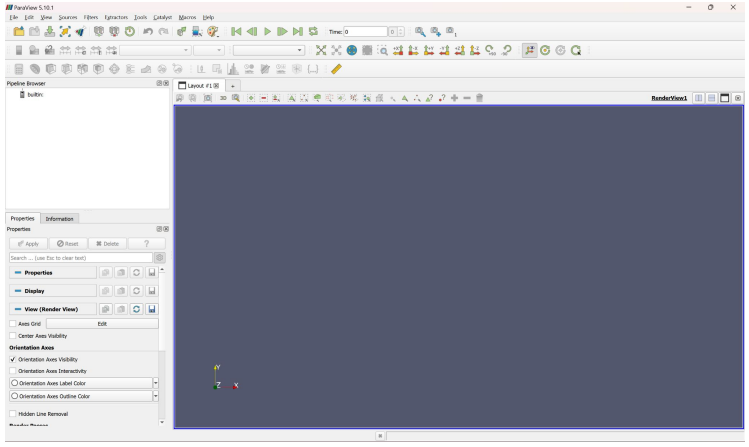
Using paraview to look at CFD simulations!

1. Click the paraview symbol on your computer's task bar



2. You should see something that looks like this:

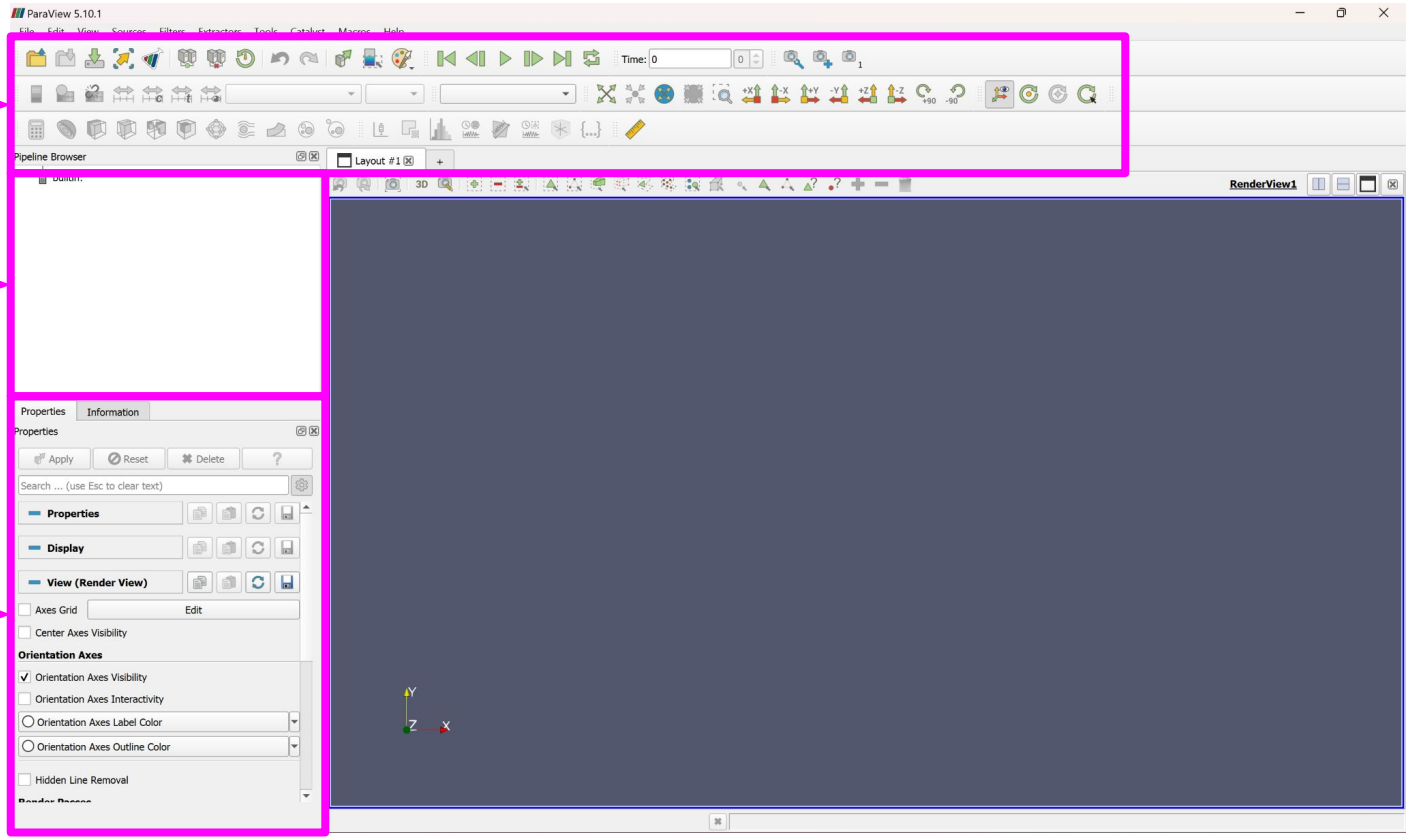


Using paraview to look at CFD simulations!

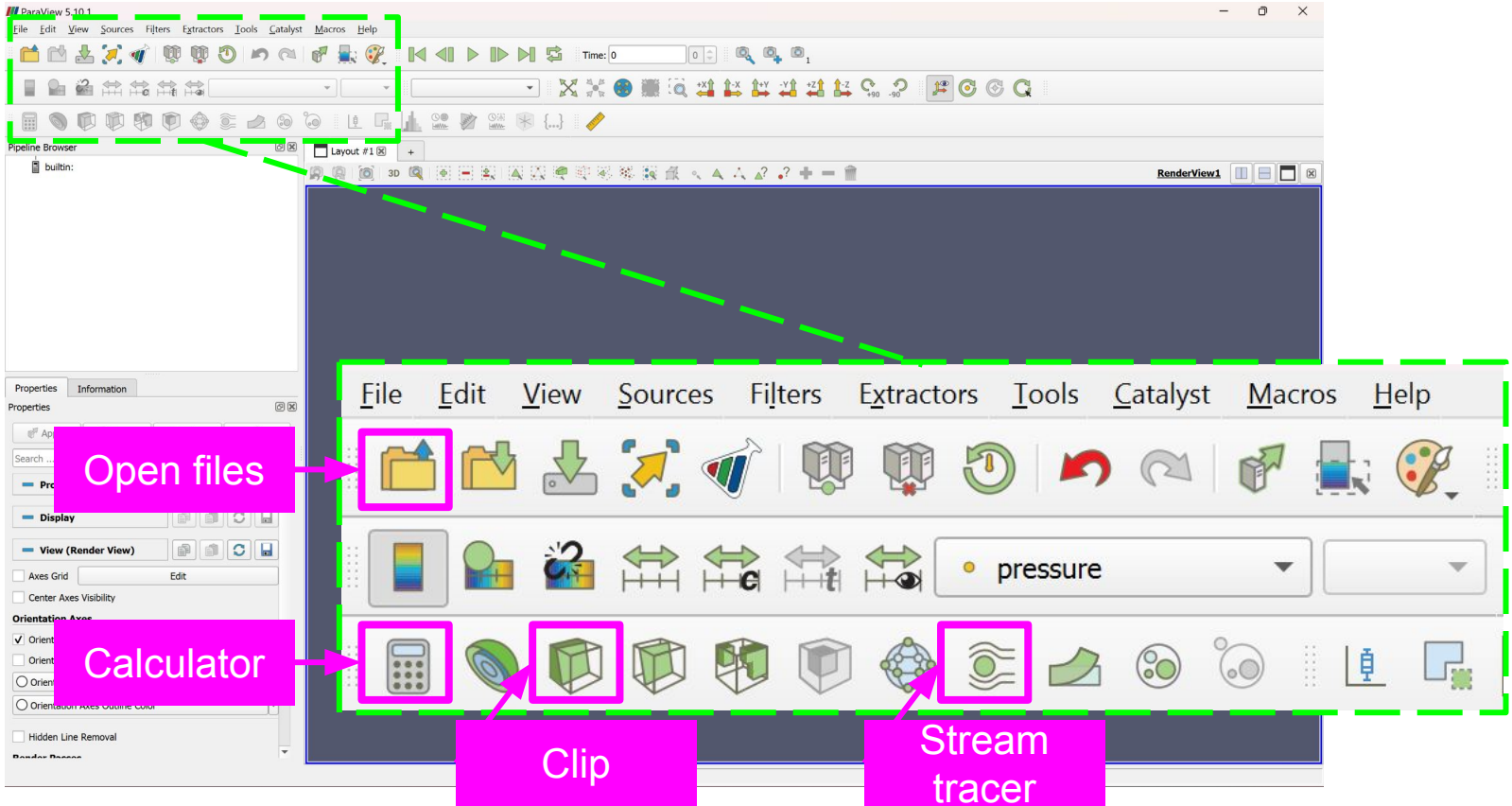
Buttons for common tools we will use

Pipeline browser: where our aorta models and modifications are

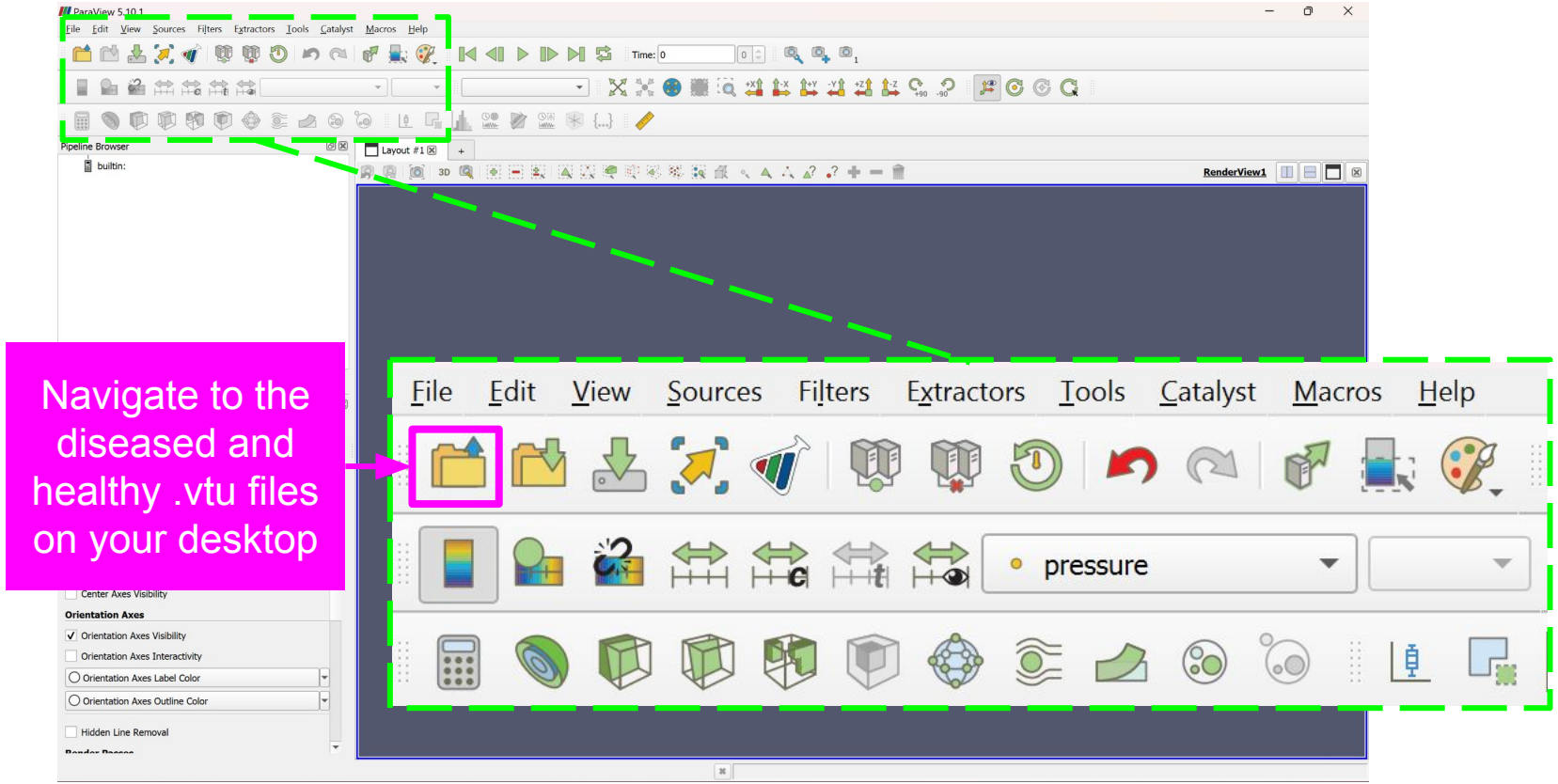
Properties: where we will control how we change our aorta model



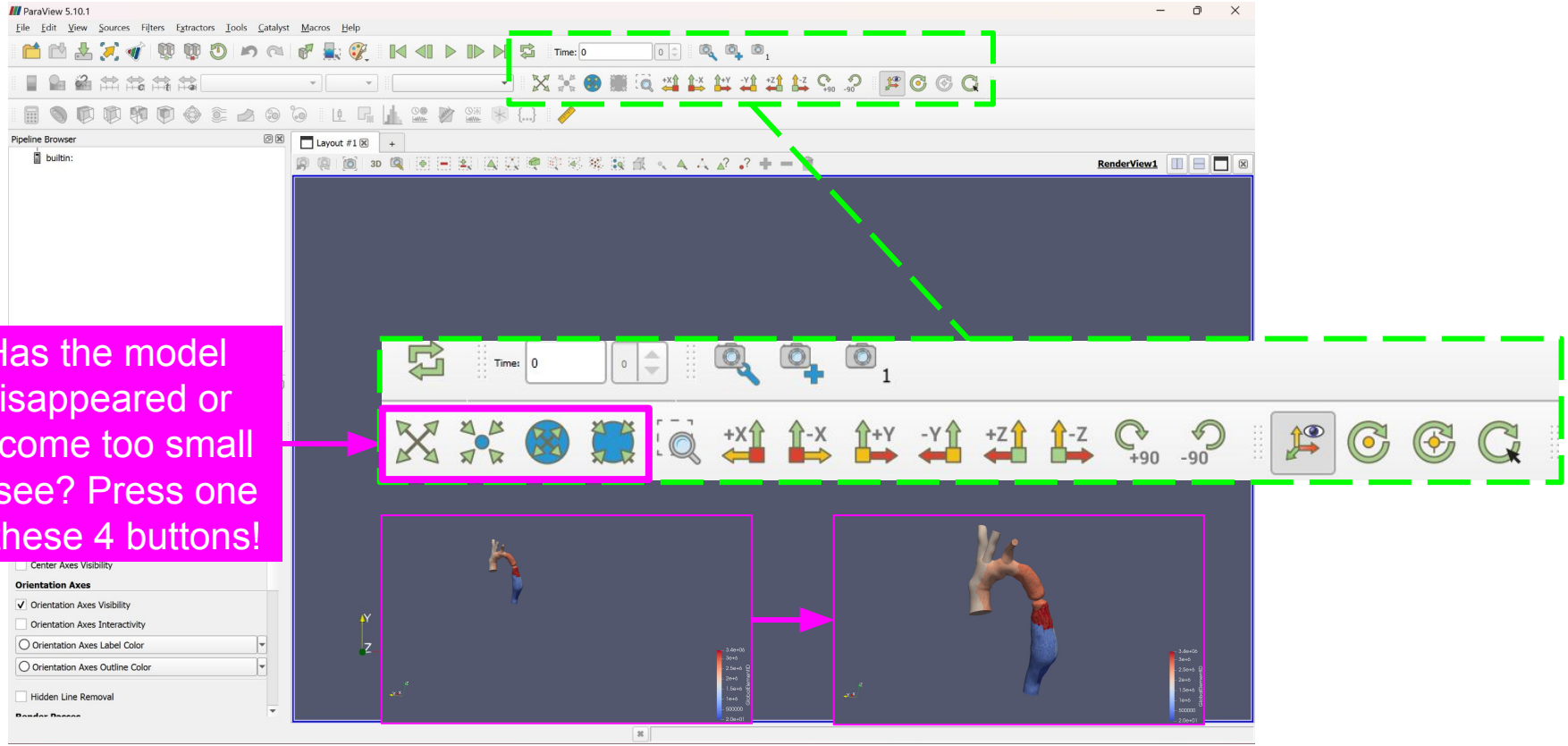
Important buttons in Paraview



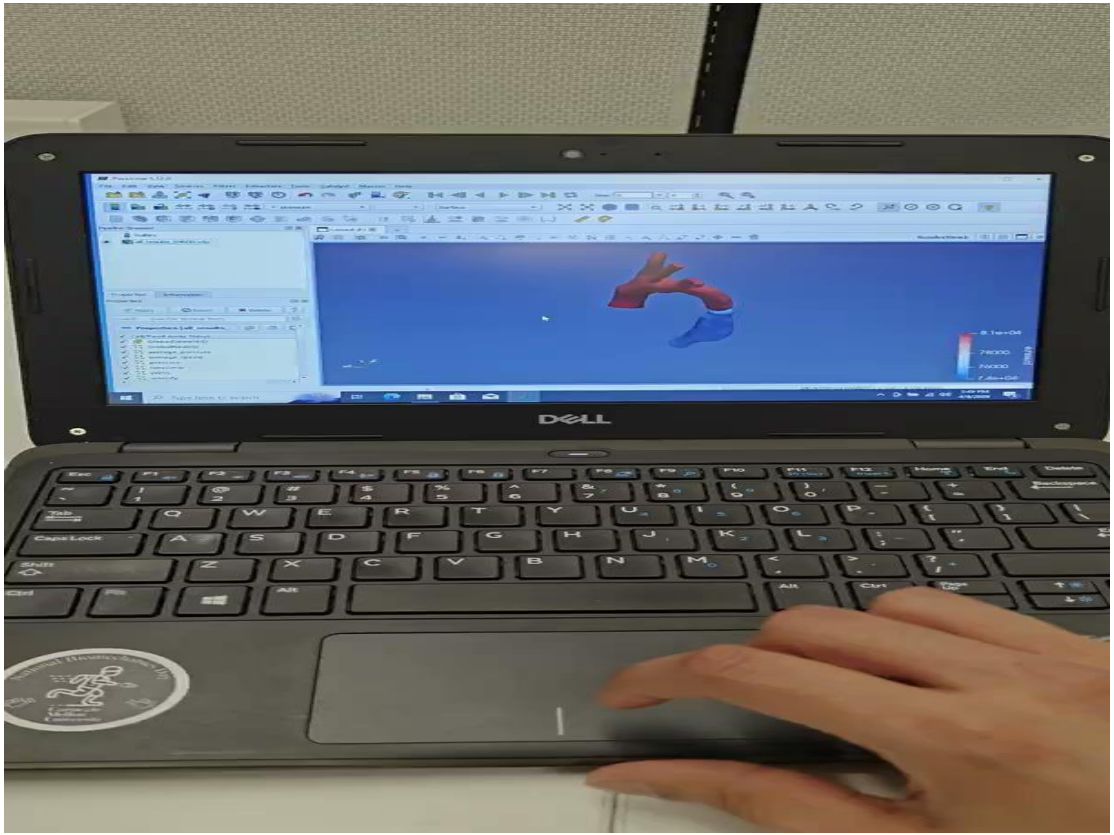
Loading data into Paraview



Zooming back into your aorta model

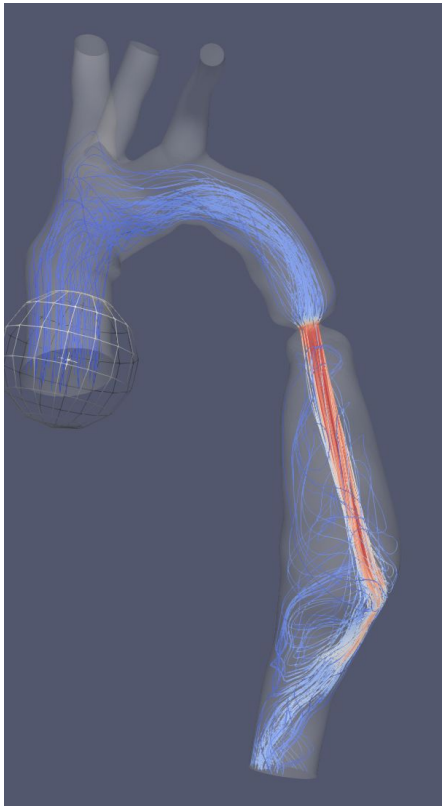


Zooming into your data and rotating it

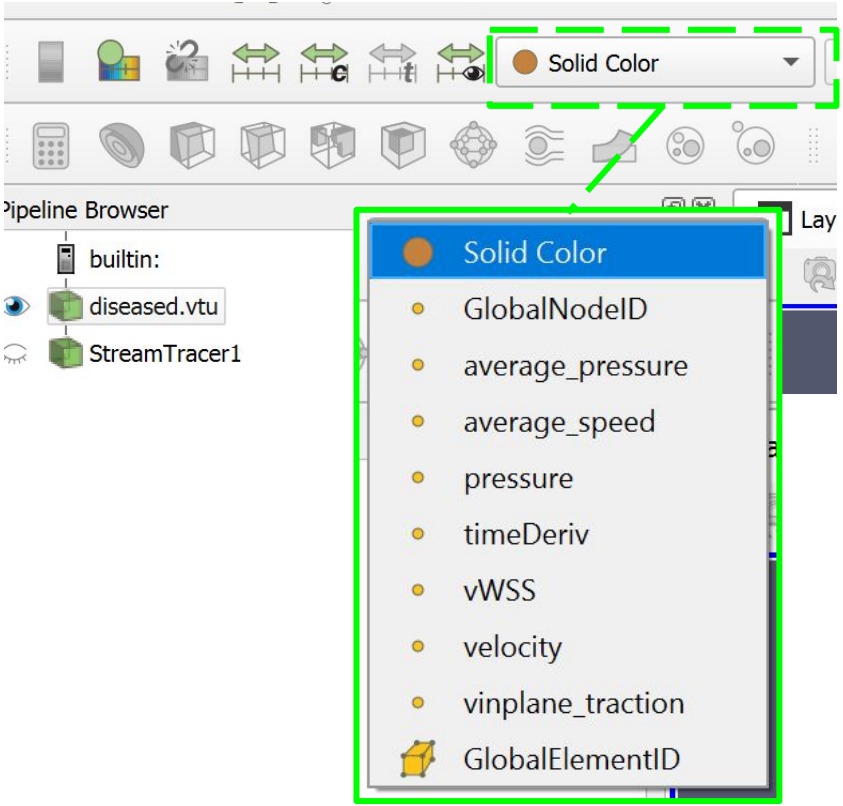


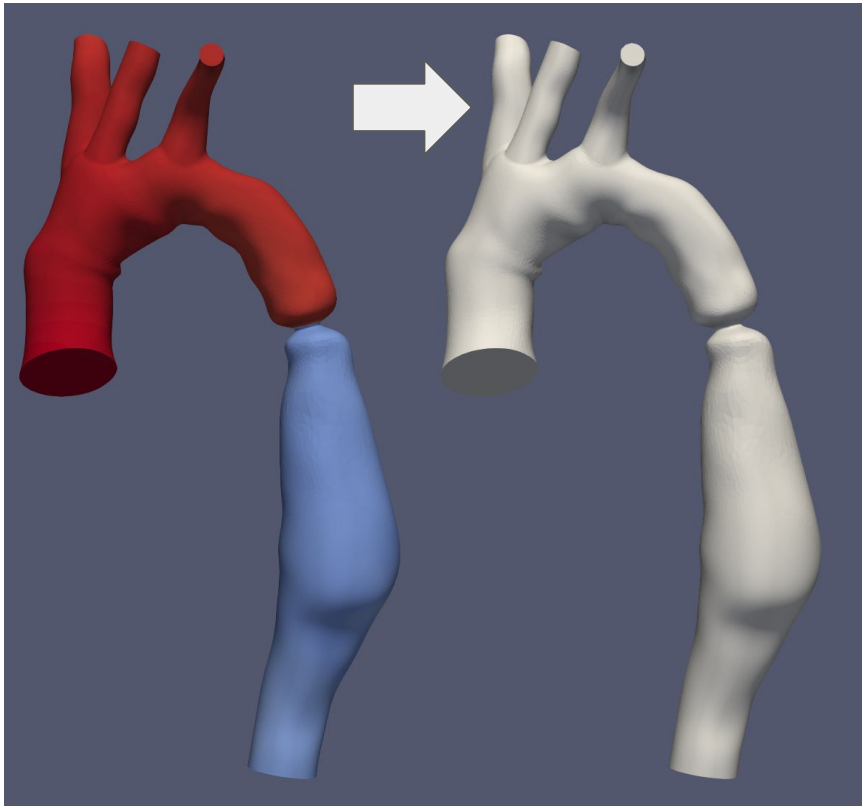
Viewing Streamlines

Looking at streamlines tells us a lot about the flow! But getting our software to show this takes some steps.

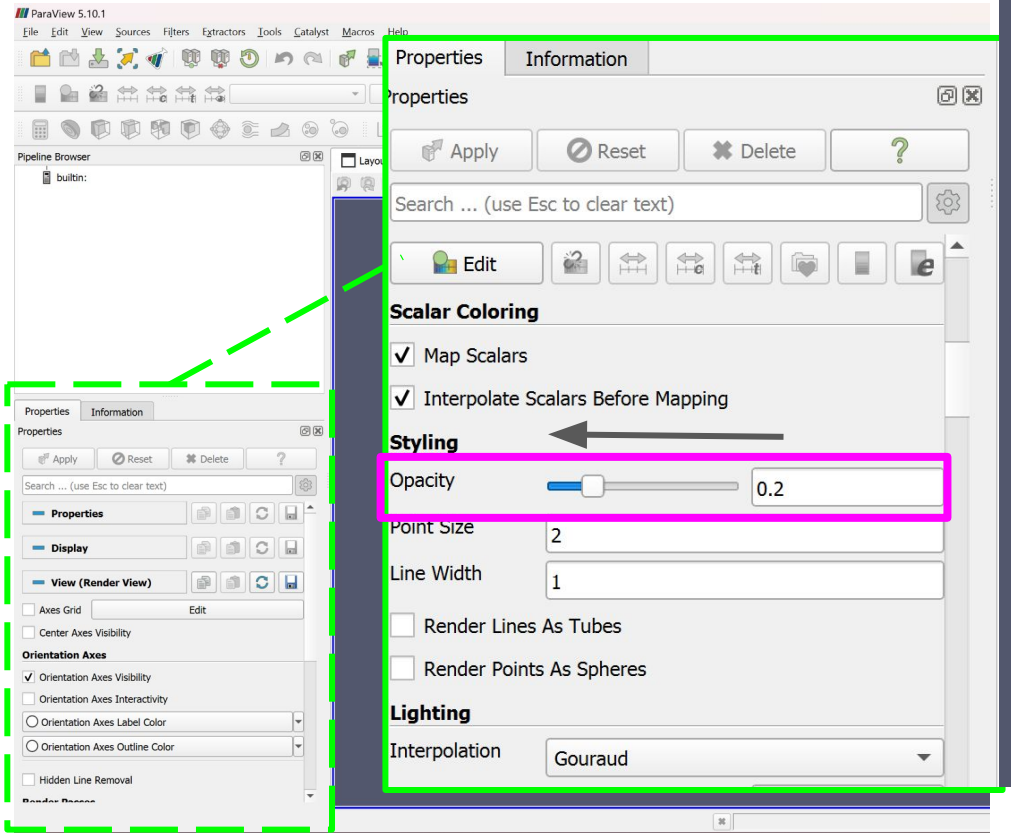


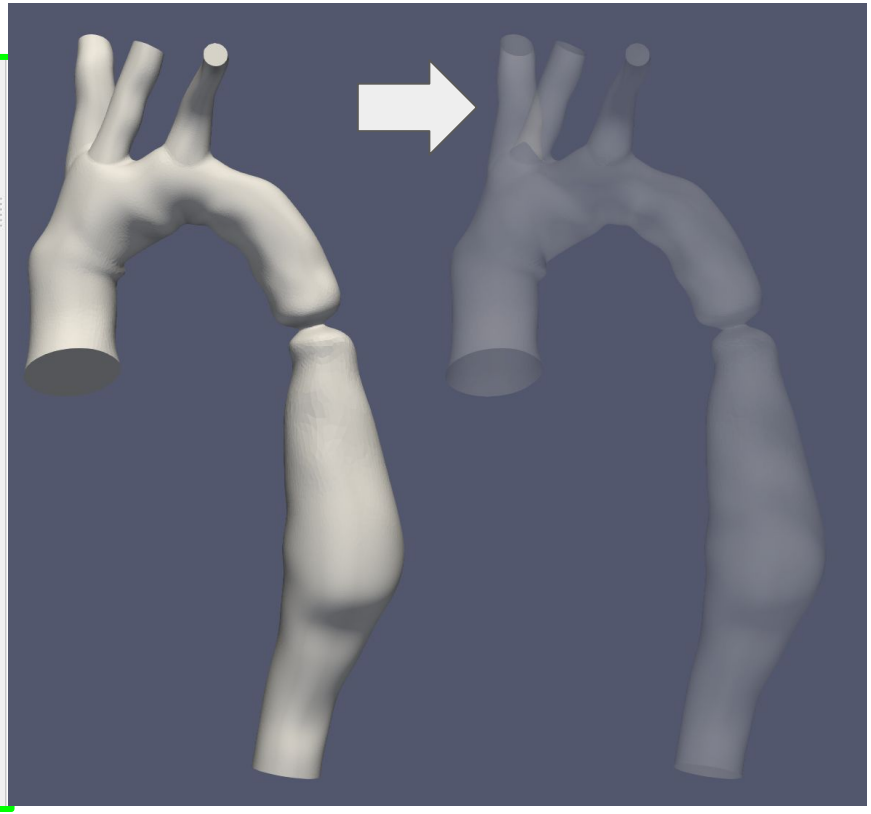
1. change to solid color



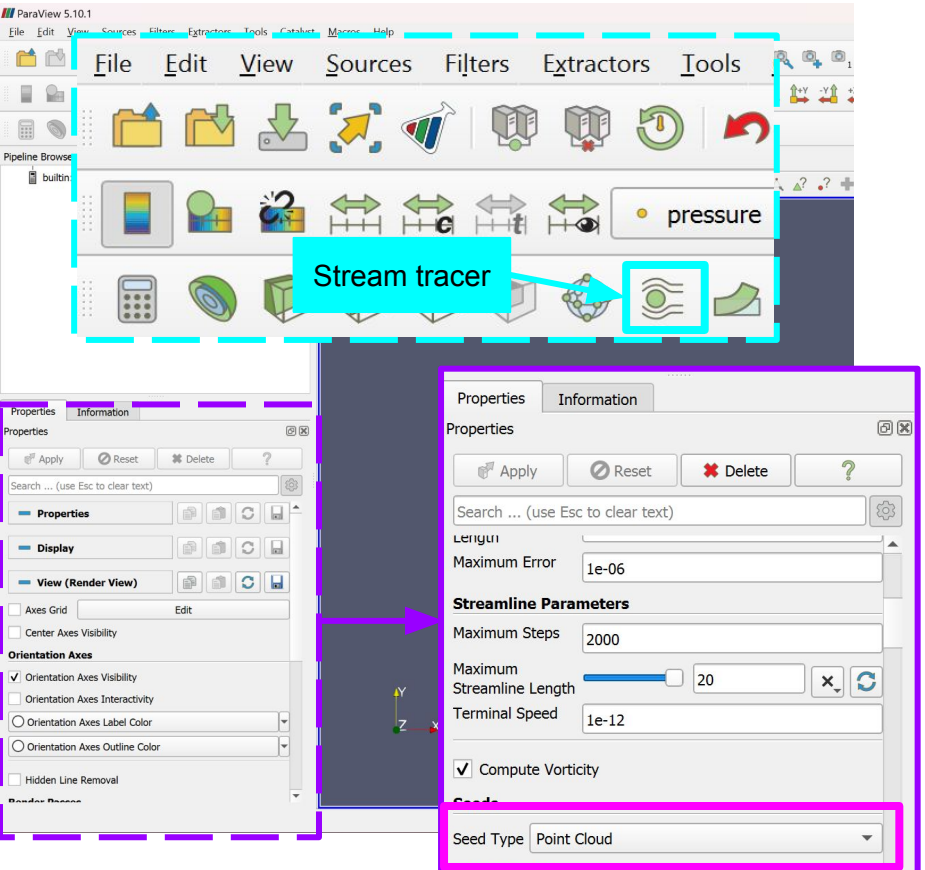


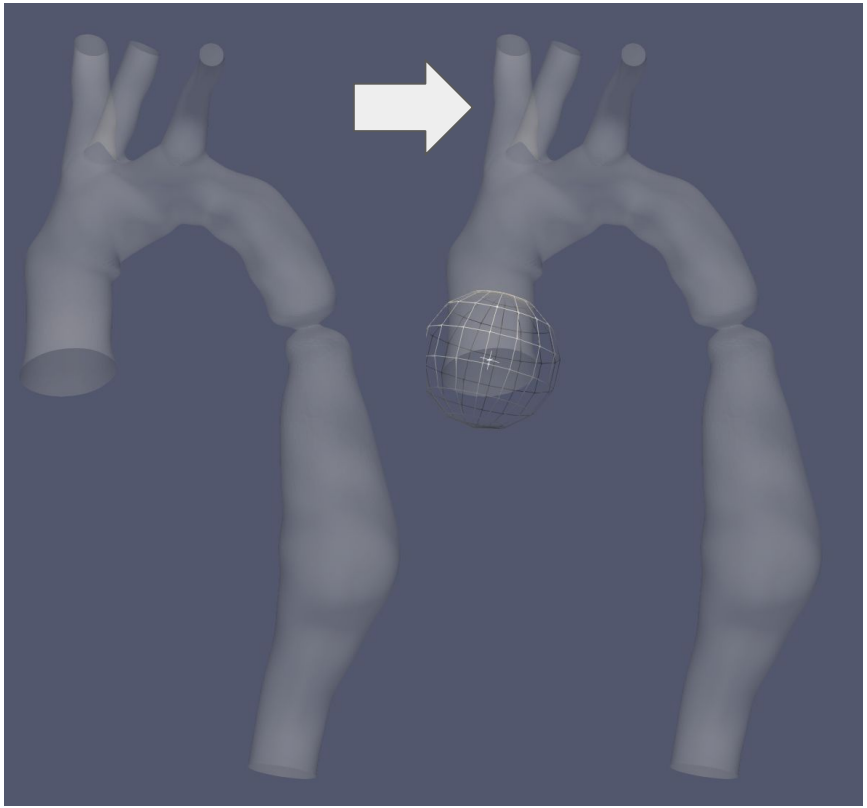
2. Change opacity to 0.20





3. Click stream tracer and set seed type to point cloud





4. Change some settings for the streamlines, then apply!