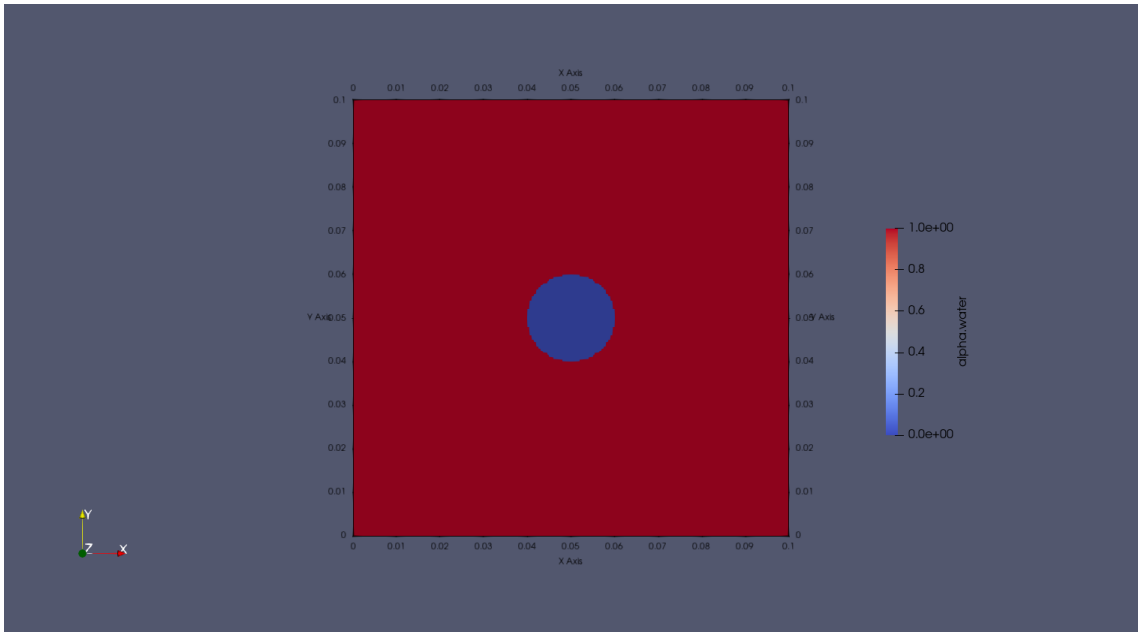
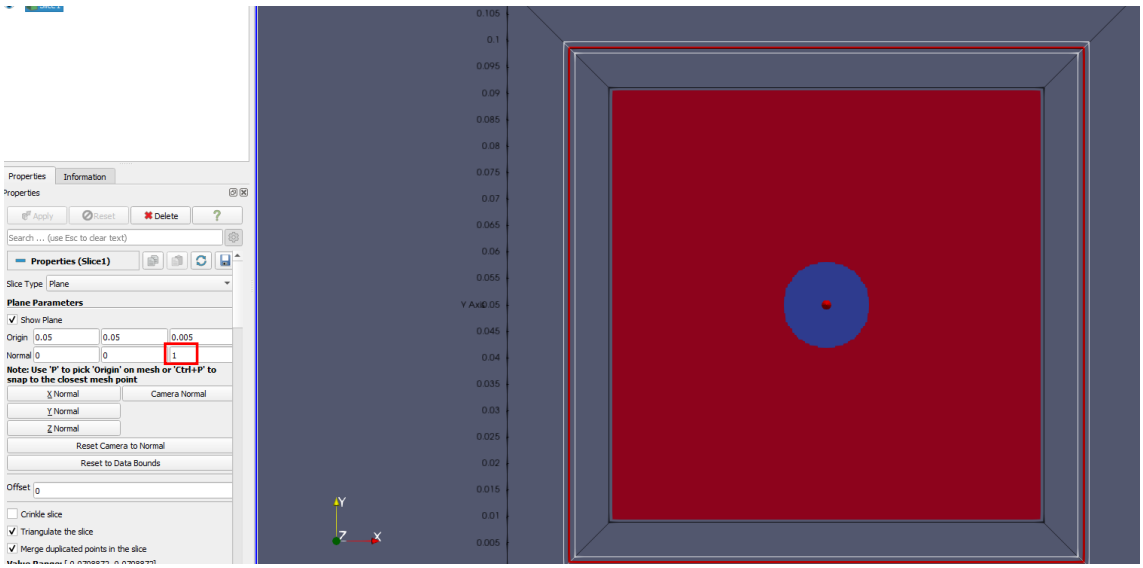
How to calculate an area using ParaView:

Here it is displayed the “alpha.water” fields at some random moment.



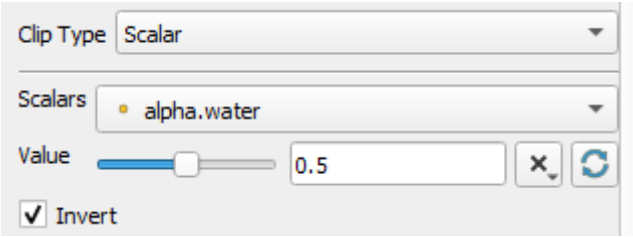
Although our simulation is 2D remember OpenFOAM uses a one cell on the third dimension so we need first to slice our domain in a plane. Lets use the tool “Slice”  and impose the normal of the plane to be pointing in our third dimensions. In my case, the third dimension is Z.



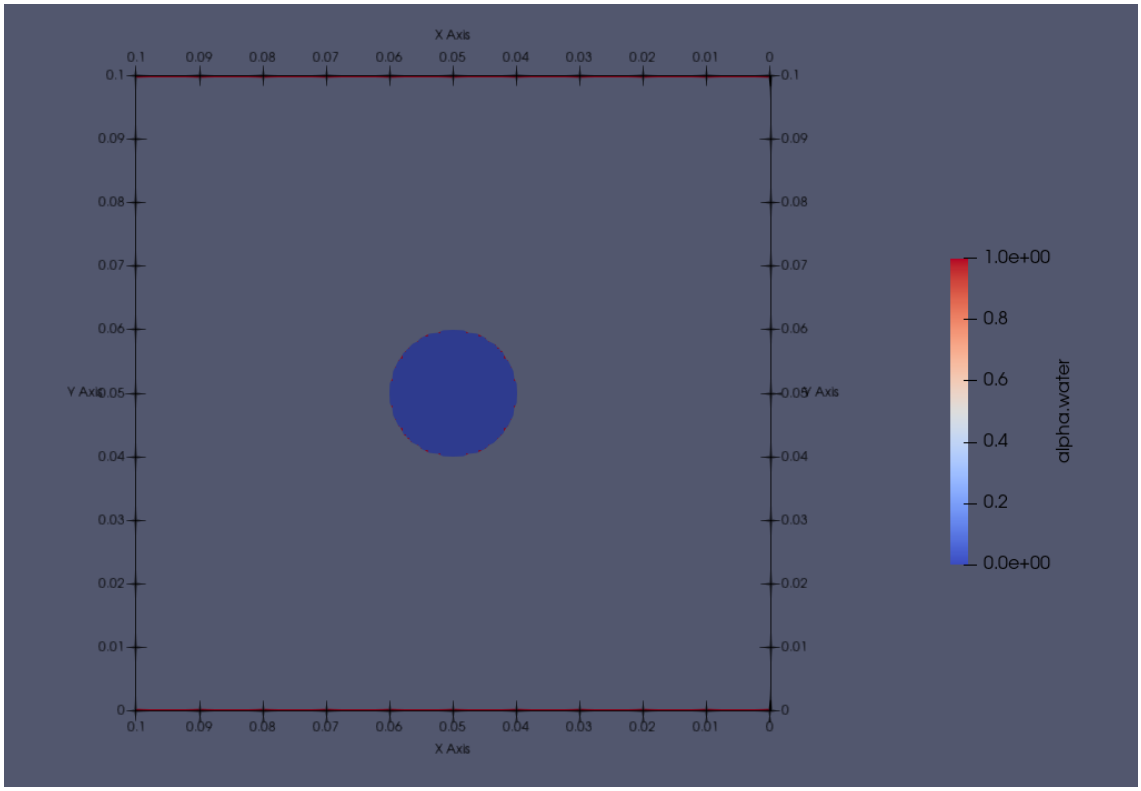
Once this is done, lets display the region where we want to calculate the area. In this case we want to calculate the area of the air bubble, thus, the alpha.water values <0.5.

To do so, we will use the tool “Clip”  set the cliop type to “Scalar” and in scalars select the field “alpha.water”.

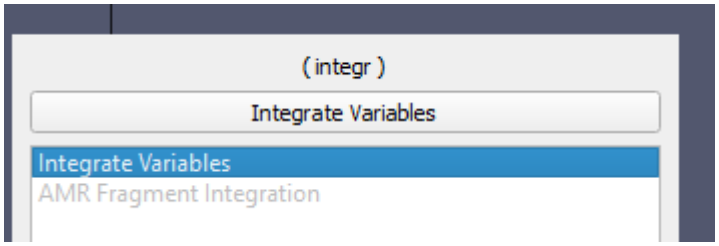
Finally select a value of 0.5.

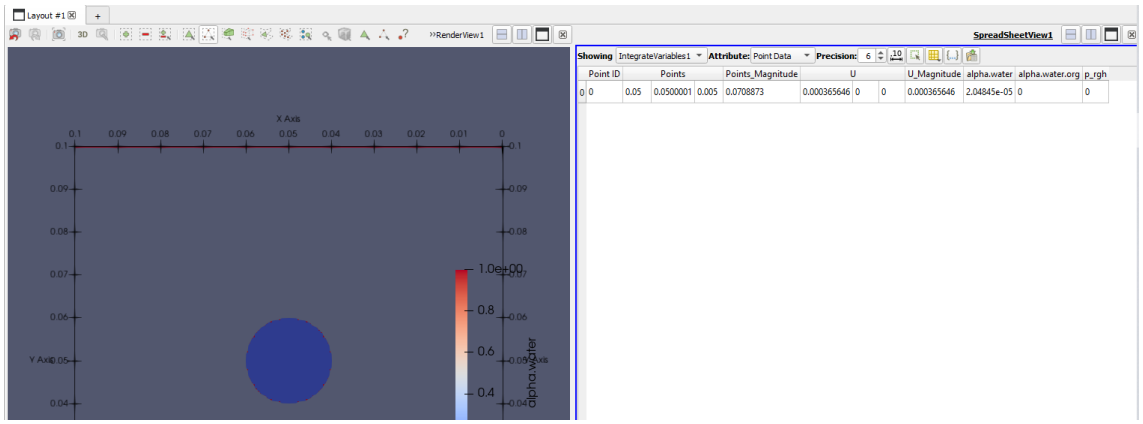


And we shall see something like this:

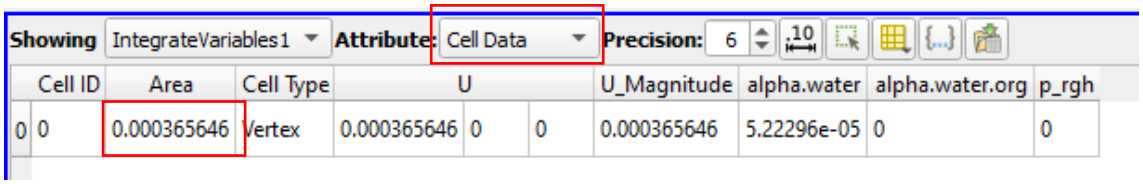


Finally we “Search” for the tool “integrate Variables” and this will open a new window behind our results.





There, if we select the attribute “Cell Data” it will give us the Area at the corrent time step.



Showing IntegrateVariables1 Attribute: Cell Data Precision: 6

	Cell ID	Area	Cell Type	U			U_Magnitude	alpha.water	alpha.water.org	p_rgh
0	0	0.000365646	Vertex	0.000365646	0	0	0.000365646	5.22296e-05	0	0