Tutorial: How to use ParaView to show CellID and cell center data

by Xiaofeng Liu, Ph.D., P.E.
Assistant Professor
Department of Civil and Environmental Engineering
Penn State University

In computational modeling, it is sometime desired to inspect the value of solution variables at a cell or point. For example, during debugging process. The purpose of this tutorial is to demonstrate the steps to display cell IDs and cell center values using ParaView. There are of course other alternative ways to do it.

1 Steps to display the cell ID and cell values

- Launch ParaView and load the OpenFOAM® simulation case as usual (Figure 1).

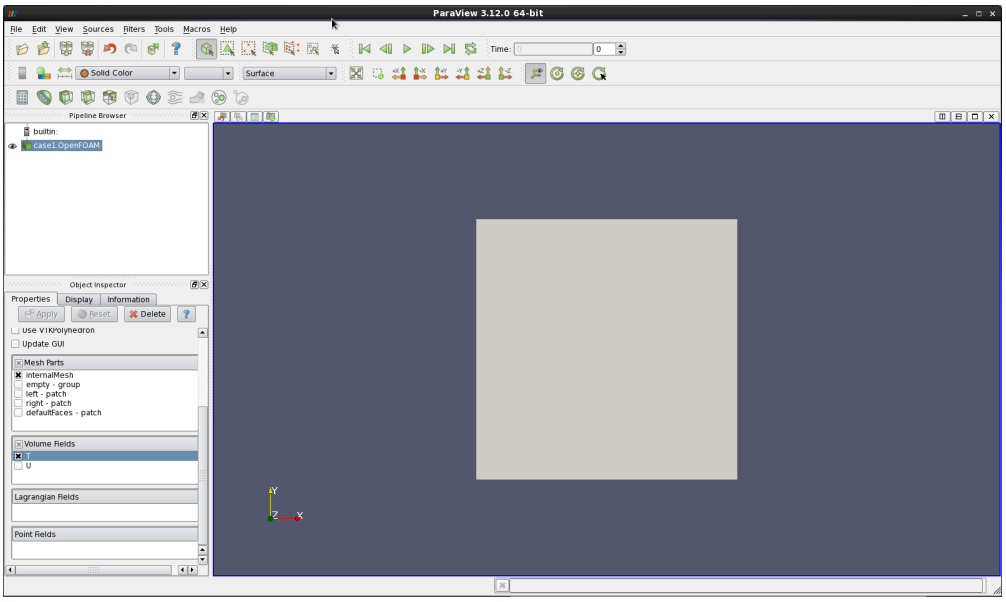

Figure 1: Loading of mesh and data using *ParaView*

- Navigate to the desired time step and plot the surface with edges (for the purpose of seeing each cell). Plot the contour of the variable using the cell center values (Figure 2).

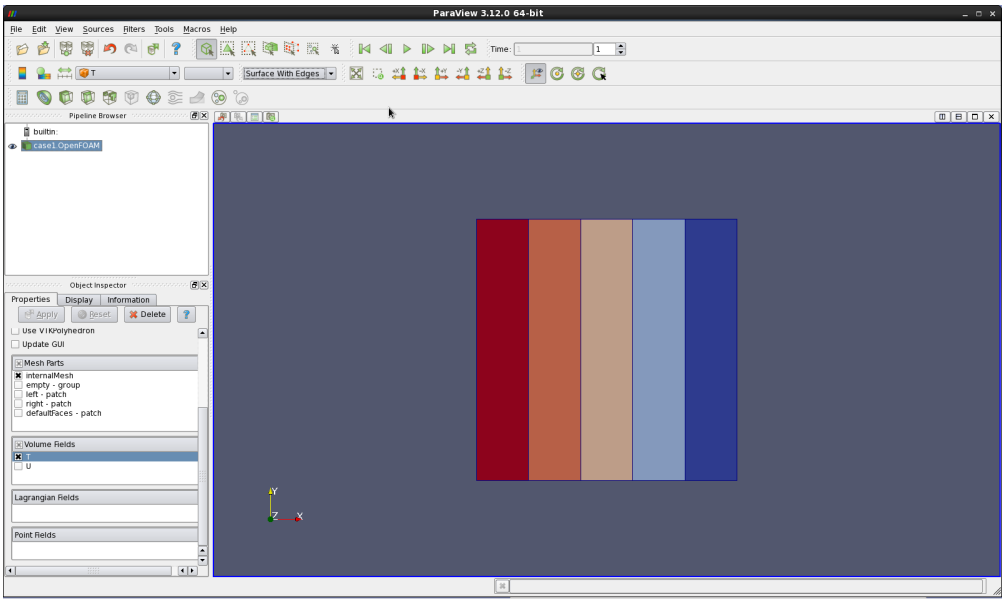

Figure 2: Plot contours of the variables using cell center values

- Create another view window and select the "*Spreadsheet View*" (Figure 3).

- Change the *Attribute* to *Cell Data*. Note the "Cell ID" starts from 0, which is conformal to the notation in OpenFOAM® (Figure 4).

- Click on any line in the "*Spreadsheet View*" will highlight the corresponding cell in the 3D view window (Figure 6).

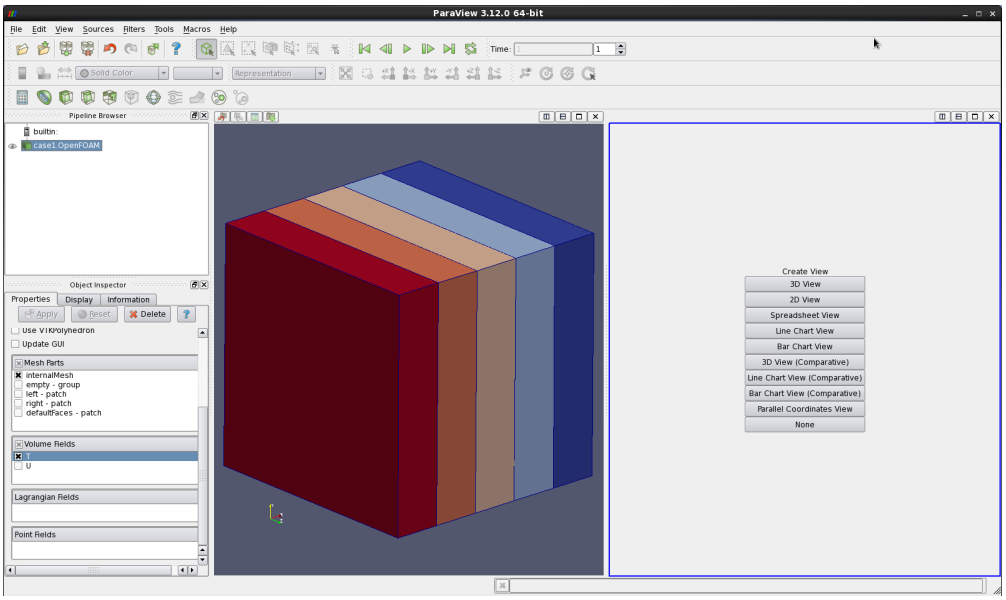

Figure 3: A new view window selecting "*Spreadsheet View*"

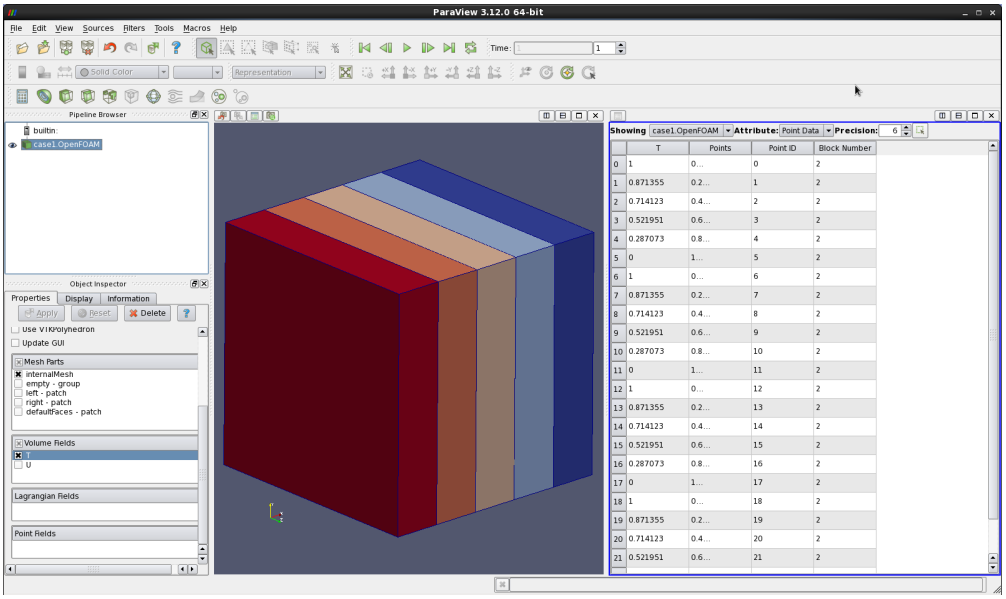

Figure 4: "*Spreadsheet View*" with the default *Attribute* of "*Point Data*".

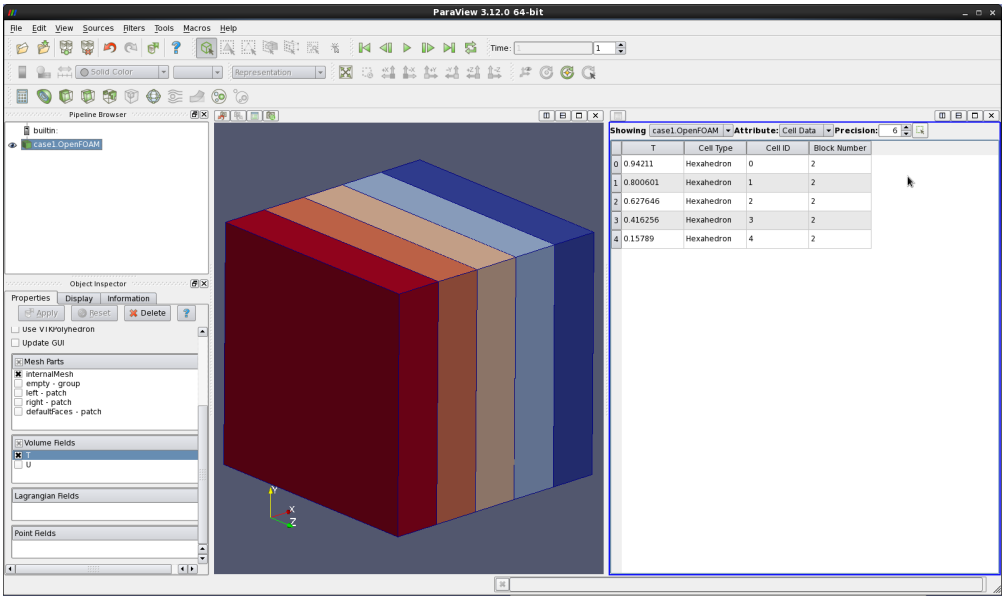

Figure 5: "*Spreadsheet View*" with the new *Attribute* of "*Cell Data*".

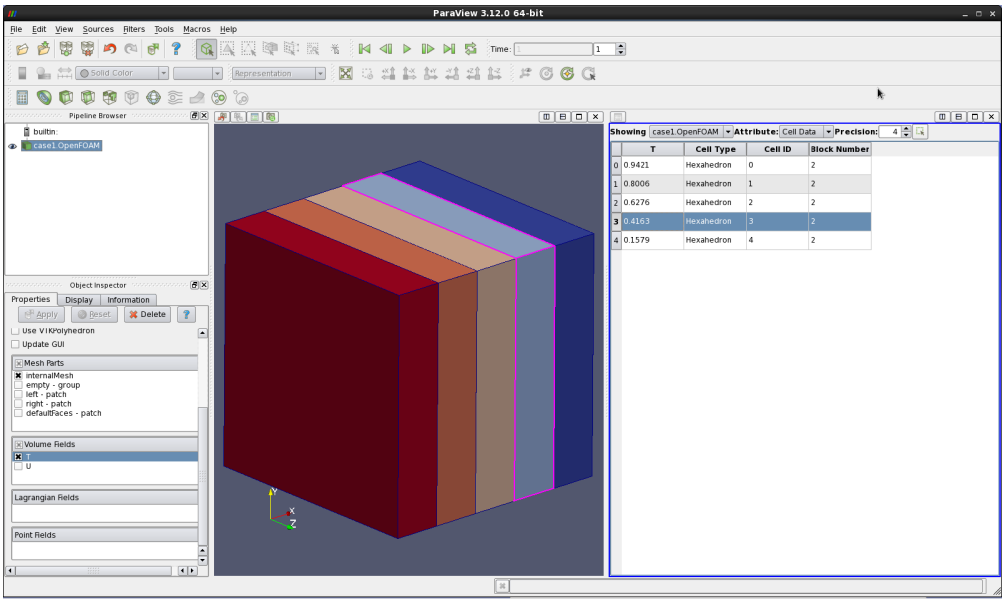

Figure 6: Highlight individual cells by clicking lines in *Spreadsheet View*.