ParaView Viewer User Guide
[version 4.0.1 Ubuntu]

Jeffrey Chen
Spring 2015

Introduction:
ParaView is an open-source, multi-platform data analysis and visualization application. ParaView users can quickly build visualizations to analyze their data using qualitative and quantitative techniques. The data exploration can be done interactively in 3D or programmatically using ParaView's batch processing capabilities.

Linux package:
If you are using a Linux machine the best way to install ParaView is through the store. This is best because it includes add-ons that you need for Meep and will save you a lot of time since you will not have to extract the packages separately.

Add-ons included:
- HDF5 files visualization tools(h5utils)
- Hierarchical Data Format 5 (HDF5) – Runtime tools (hdf5-tools)
- ParaView Viewer

There is a very great tutorial guide for the program but because we will be mainly simulating waves, much of the performance of ParaView will not be used. So it is more important in our case to learn the basics and how to bring a simulation into ParaView rather than to make edits and render the image in unique ways although this is possible because Meep will produce the simulated files into an array of values. Of course ParaView is an excellent program so it will definitely not hurt you to learn how to use it for modeling.

ParaView and Meep:

ParaView works best with programs such as Python, because you can actually manually adjust the code in which you can directly apply it to your visual. But because Meep does not have a relationship like that, we will be using more of an old school approach to view our simulations. The idea here is to save our Simulations as an .h5 file and then open it in ParaView. The upside to this is that you will not need to know how to run the programs in parallel with each other. While the down side is that every time you want to make an edit you must save a new file and open it in ParaView again. Lucky this is not hard, and as long as you have your code ready to run from copy and paste this should be an efficient way to do this.

ParaView basics:

Below is a description of what some of the buttons do in ParaView. Again this is a very powerful visualization tool and has potential to create almost any kind of visual adjustment to our simulation. But because we will be mostly dealing with two-denominational waveguides a big portion of the program will not be used.

Pipeline Browser:
After you have loaded in your simulation file, which you simply just go to ~\file\open*Your_File.h5*
your file will then open up under your pipeline browser on the left hand side.

Pipeline Example:
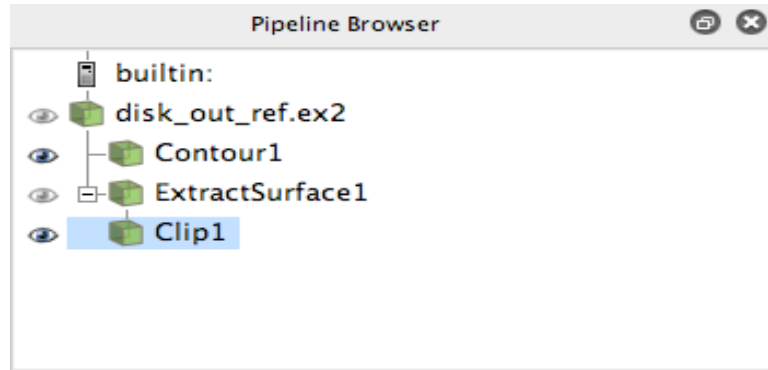
* Even though you have just loaded in your file, it does not mean that it will display on your map. You
must now click on your file and hit the "Apply" button below.

At this point you should have your simulation on the screen. The next important piece is knowing how
to run your simulation. Lucky ParaView makes this very straight forward and simple. At the top of
your tool bar you will see a set of switches to play, fast-forward, loop, etc. Simply click play to run,
and make your adjustments as needed with the other tools.
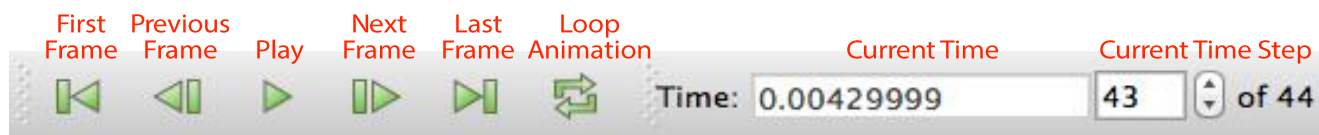
Toolbars:

The last important piece is the Toolbar, for the most part you will not need to know what all of the
buttons do. But it will be nice to click on the different settings to adjust your simulation. One of the
very nice features is that you can adjust your color outputs and surface textures. This will allow you to
see your projects in different visuals which can be a very great tool at times.

Toolbars
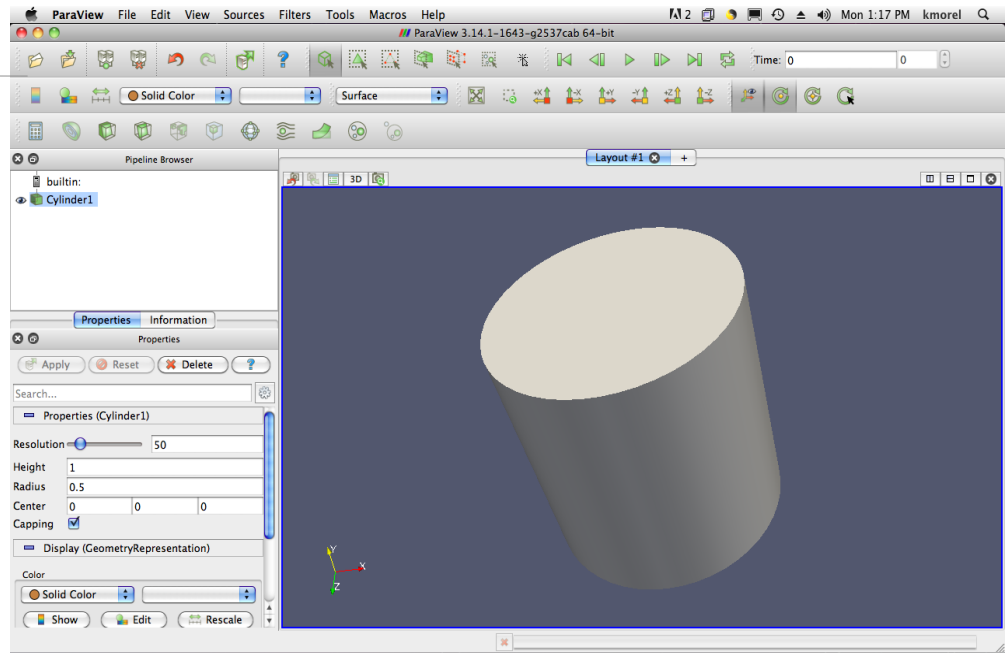
Camera Manipulation (Toolbar):

Below is a visual of the camera setting buttons on the Toolbar, you can use these different settings to adjust your viewing visual.