

# Visualizing Output With Paraview

April 12, 2016

# Chapter 1

## Paraview

ICE features functionality for visualizing models using ParaView.

### 1.1 Installation and Configuration

ParaView use for ICE requires a Mac OS or Linux operating system, as ICE does not currently support ParaView connections on Windows. You will also need an installation of ParaView on your local machine. ParaView can be downloaded from its official website. The ICE development team recommends using the latest available version of ParaView, currently 5.0.1 at the time of this writing. You will further need a custom Python HTTP web server implementation, which can be downloaded from the Oak Ridge website.

#### 1.1.1 Launching ParaView

Before connecting to ParaView, it must be launched from the command line. This can be done by running the command:

```
DISPLAY:0 (ParaView installation folder)/bin/pvpython (Path to web server file)/http_pvweb_server.py --host localhost --port 8600.
```

On Mac, the first path should instead end in `/paraview.app/Contents/bin/pvpython` to specify the file inside of the ParaView application.

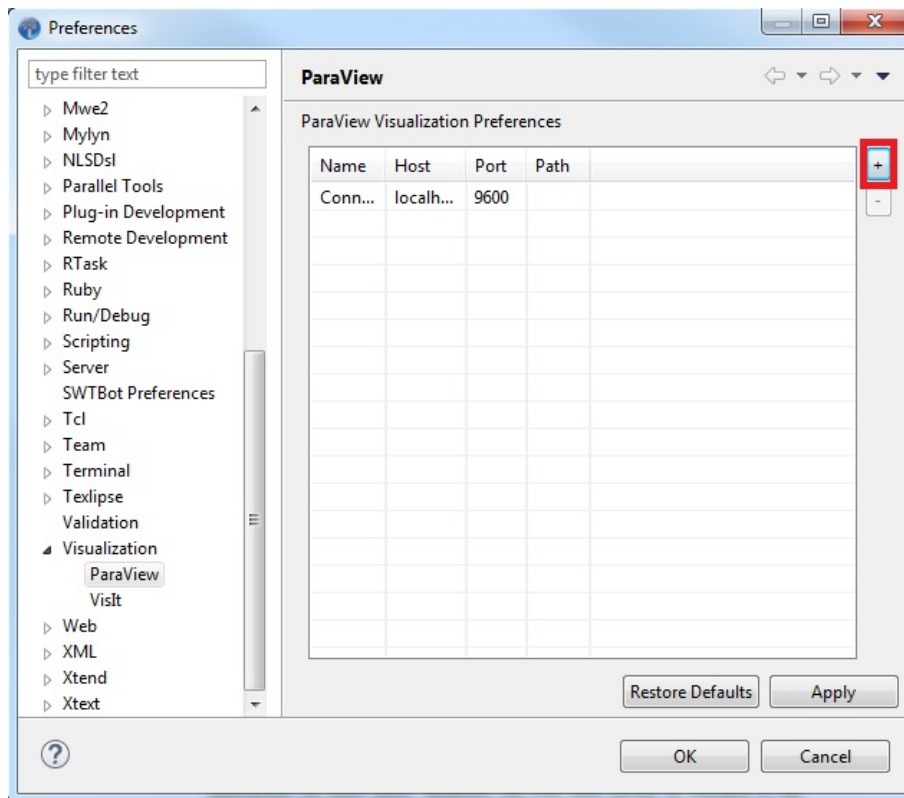
This will launch the web server script in ParaView Python. You can specify

a different port as needed by using a different number after “-port”.

### 1.1.2 Configuring the ParaView Connection

Once ParaView is running, ICE must be configured to connect to the server. This is done through specifying a default connection in the ICE Preferences page. This process only needs to be performed once. After initially creating the connection, ICE will attempt to connect to ParaView on that port each time it is launched.

To set the connection, select Window → Preferences... in ICE’s menu bar. (On Mac OS X, Preferences... is located under ICE instead of Windows.) Select Visualization → ParaView in the tree on the left side of the Preferences window.



Press the button with a “+” symbol in the upper right (highlighted in the image above) to add a new row to the table. Click on cells in the new row to edit their values. The default values automatically supplied by ICE should be

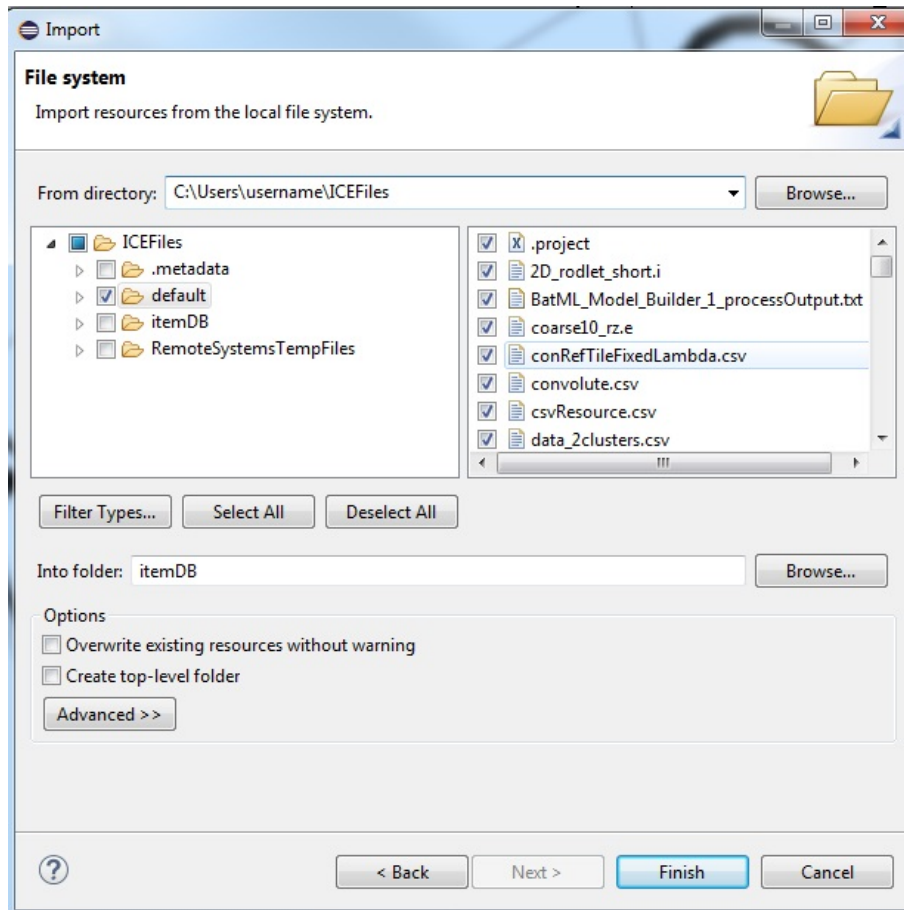
appropriate for most users. However, the Port field should be changed to the port number you specified when starting the server. (8600 if using the example above.)

Once finished editing the cells in the new row, press Apply, then OK. ICE will then connect to the ParaView server.

## 1.2 Opening a ParaView File

To open a ParaView Plot Editor, a file that uses this editor must first be placed in the Project Explorer. This view lists files imported into ICE. To access the Project Explorer, use the the menu bar at the top of the window and navigate to Window → Show View → Project Explorer. Depending on the active Eclipse perspective, opening this view may require selecting Other... and finding the Project Explorer in the dialog under the General category in the tree.

By default, the Project Explorer should automatically import the ICE-Files/default and ICEFiles/itemDB folders. If it does not, or if you want to import a different folder into ICE, right click in the Project Explorer and select Import... from the context menu. Then, select General → File System from the tree, and press the Next button. Select directories and/or files to import into the Project Explorer, and enter which folder they should be imported into, as shown below.



Once a file is in the Project Explorer, simply double click on it to open it in ParaView.

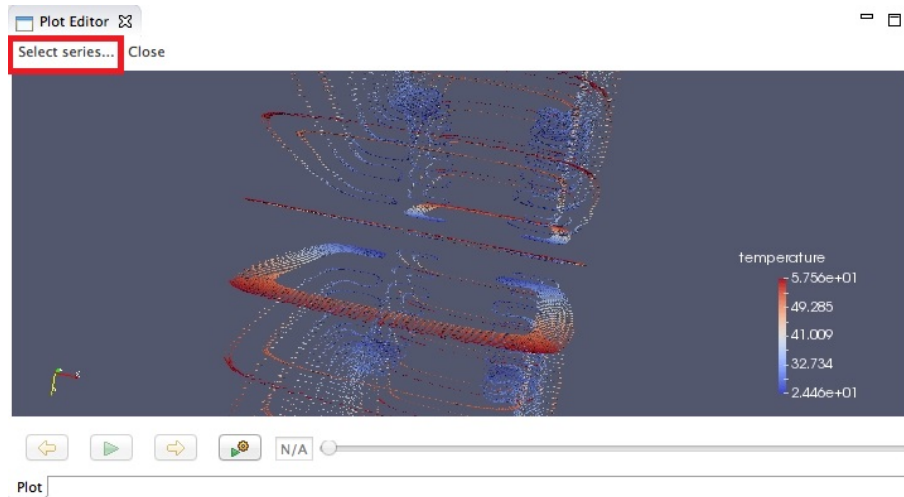
### 1.2.1 Using ParaView

#### Camera Controls

The Plot Editor allows the user to rotate the model by clicking and dragging inside the display area or adjust the zoom by scrolling the mouse wheel. Other commands vary slightly between the two utilities.

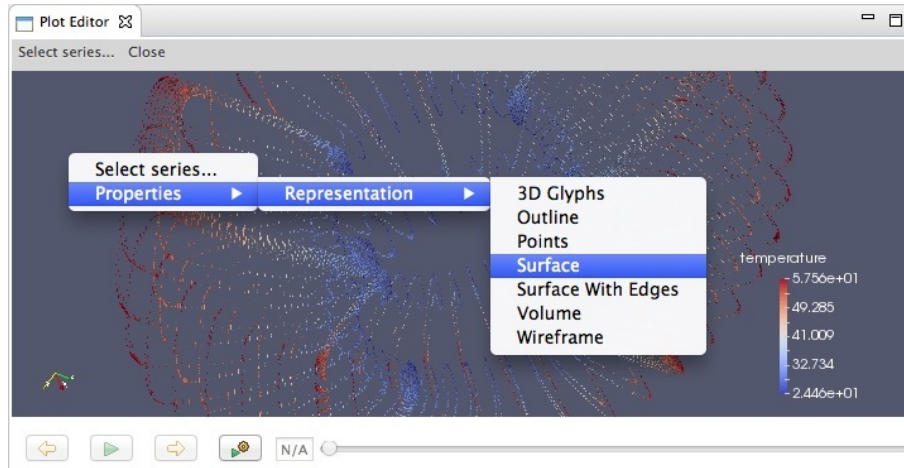
## Selecting the Plot

Pressing the Select Series... button will open a dialog which lists the various plots in the opened file. Simply select one and click OK to open it.



## Setting the Plot Representation

ParaView is capable of displaying plots in several different representations, such as points or surfaces. To switch between plot type, right click inside the Plot Editor's display area and select one of the listed options under the Representation category in the context menu.



## Animation and Time Data

The Plot Editor features a time slider widget at the bottom of the screen.



The controls, in order of left to right, are:

- 1) Return to the previous time step.
- 2) Automatically play the plot as an animation by displaying the time steps sequentially.
- 3) Advance to the next time step.
- 4) Opens an options menu that allows the user to set the playback speed, toggle whether the animation should loop when it reaches the end, and set the plot to the first or last time step.
- 5) A display for the current time step. It can be edited to set the plot to an arbitrary time step.
- 6) A slider that shows the current time step's position on the timeline. The slider can be dragged around the timeline, setting the plot's time step accordingly.

## 1.3 Accessing a ParaView Web Server

It is also possible to access the full ParaView web viewer application inside of ICE. This allows for full access to all the web viewer's features and for remote connections to another machine hosting the ParaView session, even for Windows clients.

### 1.3.1 Launching the ParaView Web Visualizer

Before connecting, the visualizer needs to be started from the command line. ParaView's package structure is different on each operating system, necessitating a different command for each.

For the following commands, replace `paraview-install` with the path to your machine's ParaView installation and `data-folder` with the path to the top level folder under which all files you wish to visualize are kept.

For **Linux** run:

```
paraview-install/bin/pvpython paraview-install/lib/paraview-5.0/  
site-packages/paraview/web/pv_web/visualizer.py --content paraview-  
install/share/paraview-5.0/www/ --data-dir data-folder/ --port 8600
```

For **Mac OS** run:

```
paraview-install/paraview.app/Contents/bin/pvpython paraview-  
install/paraview.app/Contents/Python/paraview/web/  
pv_web_visualizer.py --content paraview-install/paraview.app/  
Contents/www/ --data-dir data-folder/ --port 8600
```

In all cases, the argument following “-port” can be changed to set the port number the server will use.

### 1.3.2 Accessing the Visualizer

Now that the visualizer is running, it can be accessed by the client machine, which may be the same machine running the server.

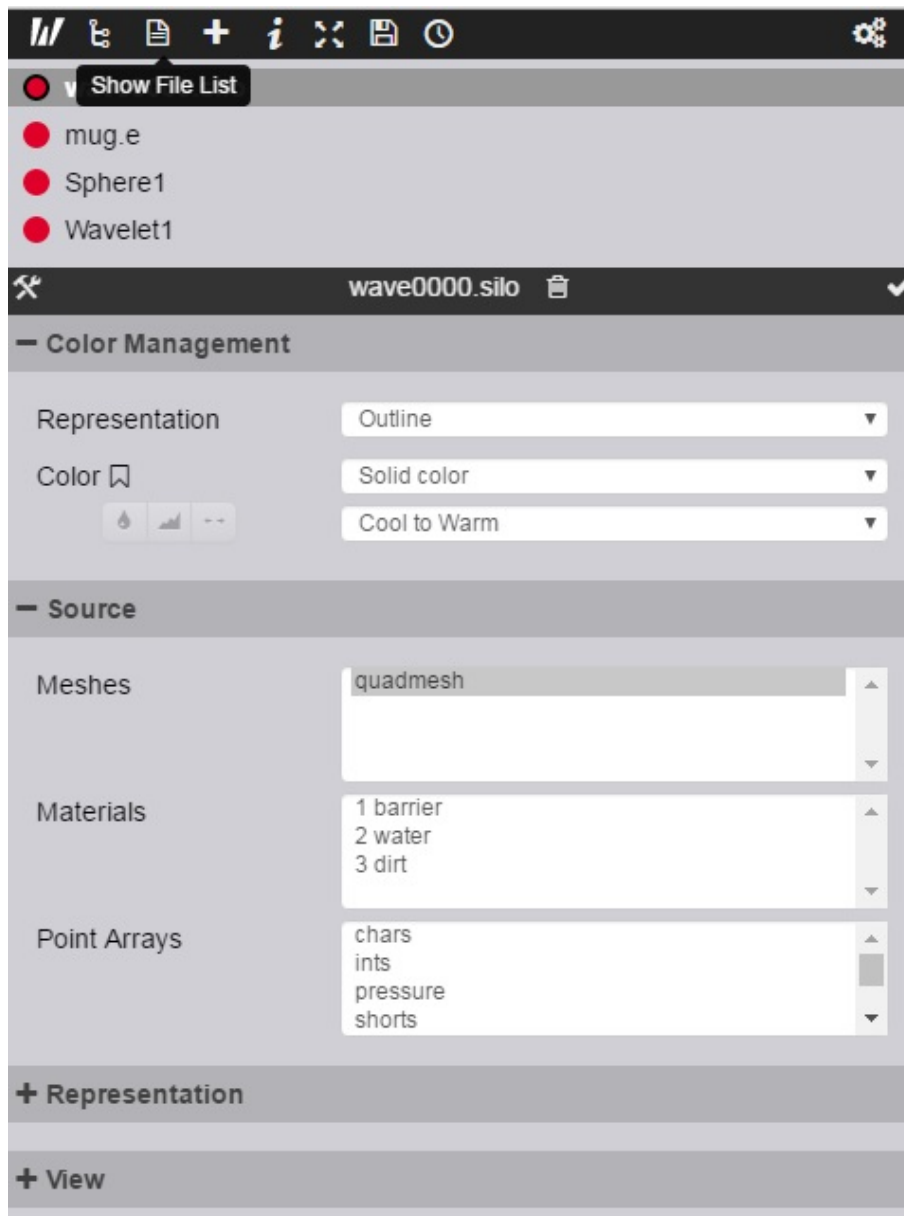
Within ICE, click `Window → Show View → Other...` to open a dialog of views. Select `General → Internal Web Browser` from it and press OK.

Finally, navigate to `http://hostname:8600/apps/Visualizer` inside the browser, replacing the hostname and port number as appropriate.

### 1.3.3 Using the Visualizer

In order to load a data file, click the Show File List button as seen below.





This will place the contents of the folder specified by the `-data-dir` argument when launching the visualizer into the side bar. You may then double click on a file to load it into the model.

You can click and drag the mouse to rotate the camera and right click followed by dragging up or down to zoom. A full description of the visualizer's

features are beyond the scope of this tutorial, but see the official documentation [here](#).