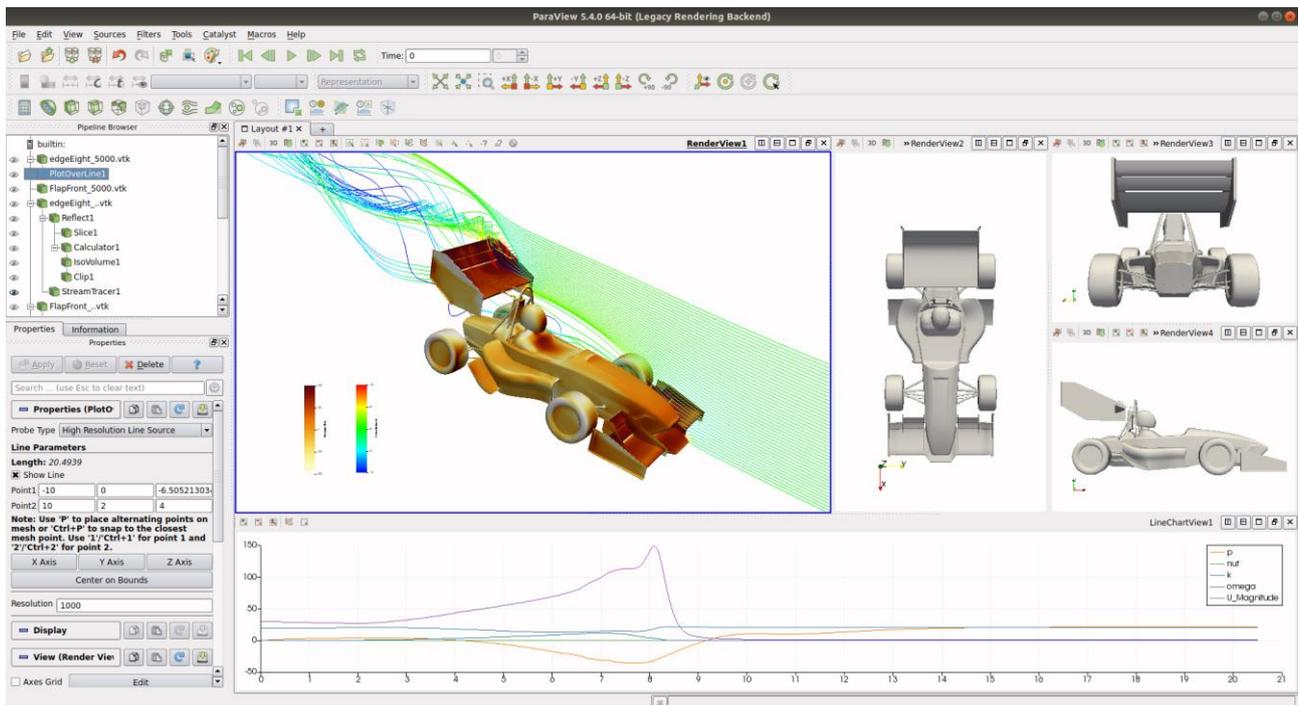


# Appendix D

## ParaView



4<sup>th</sup> edition, Jan. 2018



This offering is not approved or endorsed by ESI® Group, ESI-OpenCFD® or the OpenFOAM® Foundation, the producer of the OpenFOAM® software and owner of the OpenFOAM® trademark.

Except where otherwise noted, this work is licensed under <http://creativecommons.org/licenses/by-nc-sa/3.0/>

Editorial board:

- Bahram Haddadi
- Christian Jordan
- Michael Harasek

Contributors:

- Bahram Haddadi
- Jozsef Nagy
- Sylvia Zibuschka
- Yitong Chen

Compatibility:

- Paraview 5.0.1

Cover picture from:

- Bahram Haddadi



Except where otherwise noted, this work is licensed under <http://creativecommons.org/licenses/by-nc-sa/3.0/>

Attribution-NonCommercial-ShareAlike 3.0 Unported (CC BY-NC-SA 3.0)

This is a human-readable summary of the Legal Code (the full license).

Disclaimer

You are free:

- to Share — to copy, distribute and transmit the work
- to Remix — to adapt the work

Under the following conditions:

- Attribution — You must attribute the work in the manner specified by the author or licensor (but not in any way that suggests that they endorse you or your use of the work).
- Noncommercial — You may not use this work for commercial purposes.
- Share Alike — If you alter, transform, or build upon this work, you may distribute the resulting work only under the same or similar license to this one.

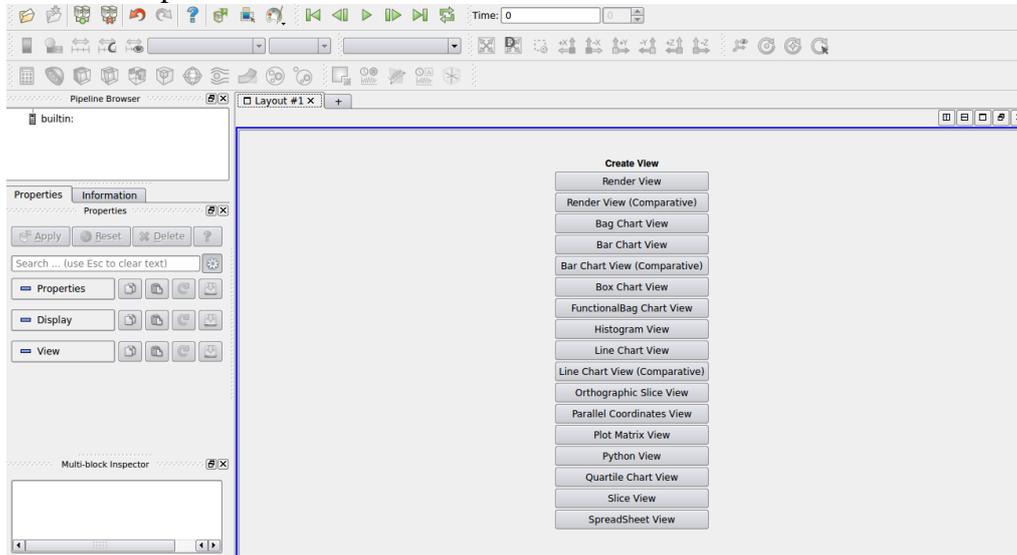
With the understanding that:

- Waiver — Any of the above conditions can be waived if you get permission from the copyright holder.
- Public Domain — Where the work or any of its elements is in the public domain under applicable law, that status is in no way affected by the license.
- Other Rights — In no way are any of the following rights affected by the license:
- Your fair dealing or fair use rights, or other applicable copyright exceptions and limitations;
- The author's moral rights;
- Rights other persons may have either in the work itself or in how the work is used, such as publicity or privacy rights.
- Notice — For any reuse or distribution, you must make clear to others the license terms of this work. The best way to do this is with a link to this web page.

**For more tutorials visit: [www.cfd.at](http://www.cfd.at)**

## 1. Introduction to ParaView

The post-processing application for OpenFOAM® is ParaView, which is a free, open source program. In this tutorial, different features and tools available in ParaView 5.0.1 will be explored.



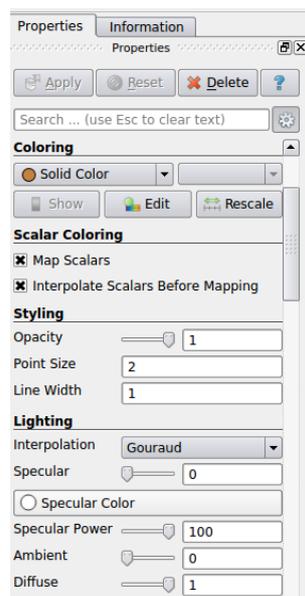
ParaView Interface

The tree structure (“pipeline”) of ParaView helps the user to easily choose and display suitable sub-models for creating the desired image or animation. Adding a mesh or velocity vectors to a contour plot of pressure is an example of this functionality.

For generation operations, use the OpenFOAM® command *foamToVTK* to convert OpenFOAM® files into readable formats for ParaView. Then open the .vtk file and press the green *Apply* button in the Properties panel. The reset button is used for resetting the window and deleting the selected operation.

## 2. ParaView Interface

### 2.1. Properties Panel



Properties panel

- **Colouring**

The drop-down menu for solid colour allows different field variables to be chosen and viewed, for example, pressure and velocity magnitude.

The Rescale button allows the data range to be adjusted to fit the data, as sometimes the max/min data range are not updated automatically.

- **Scalar Coloring**

The ‘*Map Scalars*’ option allows the scalar values to be mapped to a specific colour using a lookup table.

Turning the option ‘*Interpolate Scalars Before Mapping*’ on or off will affect the way the scalar data is visualized with colours. According to the ParaView documentation, if it is turned on, scalars will be interpolated within polygons and colour mapping will happen on a per-pixel basis; if off, color mapping occurs at polygon points and colors are interpolated, which is generally less accurate<sup>[1]</sup>.

- **Styling**

The opacity of the image can be set (1 = solid, 0 = invisible) in the *Opacity* option.

- **Lighting**

There are two options for *Interpolation*, *Gouraud* or *Flat*. With *Gouraud* shading enabled, normals are defined only per point and no face normal is needed. If the *Interpolation* is changed to *Flat*, only the face normals will be computed and used for lighting, note that this option is not suitable for objects with smooth surfaces<sup>[2]</sup>.

- **Backface Styling**

This is an advanced feature in ParaView that enables the backface style of a wire frame object to be changed.

- **Transforming**

The Transform filter allows you to translate, rotate and change the size and the origin of the data sets.

- **Miscellaneous**

By default ParaView triangulate the cells and shows them as triangles. For disabling this uncheck the “*Triangulate*” option in the Miscellaneous section of the Properties panel.

- **Glyph Parameters**

The Glyph Parameters filter generates a glyph, which can be arrow, cone, box, cylinder, line, sphere or a 2D glyph. The glyph is generated at each point in the input dataset<sup>[3]</sup>. Depending on the type of glyph chosen, different options are available to orientate, scale and size the glyph.

- **Orientation Axes**

The Orientation Axes feature controls an axes icon in the image window (e.g. to set the color of the axes labels x, y and z).

- **Lights**

The lighting controls options appear when clicking on the *Edit* button. For producing images with strong bright colors (e.g. isosurface) Headlight of strength 1 is appropriate.

- **Background**

The background color of the layout can be chosen from the drop-down menu, with types *Single color*, *Gradient* and *Image* available.

**2.2. Button toolbars**

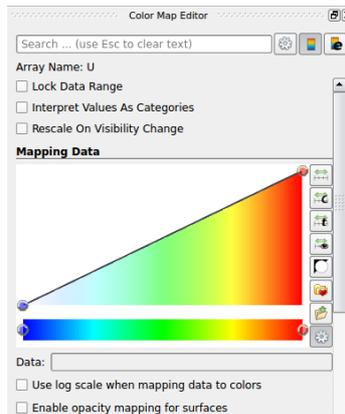


Button toolbars

Pull-down menus at the top of the main window and the major panels, in the toolbars below the main pull-down menus increase the functionality of ParaView. The function of each button can be easily understood by its icon, also any button description can be found in the Help menu (keeping the mouse over an icon without clicking on it will also give a short explanation on its functionality).

A feature worth mentioning is the drop-down menu next to the *Reset* button, this provides the options of the different ways of presenting the mesh. To see the structure of the mesh, use *Surface with Edges*; and to see both the cell structure and the interior of the mesh, use *Wireframe*.

**2.3. Color Map Editor Panel**



Color map editor

The *Choose preset* button allows the color scheme of the scale to be chosen, a common color scheme used is Blue to Red Rainbow. The *Rescale to custom range* button allows the maximum and minimum values of the color scale to be freely chosen by the user.

Another important feature can be used by clicking the button *Edit color legend properties* on the top right of the panel, this allows the scale title, font style to be changed.

### 3. Manipulating the view

#### 3.1. Contour plots

Clicking on the *Contour* button in the Button Toolbars creates a contour plot. The contour filter operates on any type of data set, but requires the input to have at least one point-centered scalar (single-component) array. The output of this filter is polygonal.

The chosen scalar field can be selected from a pull down menu. If the case is a 3D case module, the contours will be a set of 2D surfaces that represent a constant value. The *Isosurfaces* list in the Properties panel allows the user to specify the values at which the isosurfaces are computed.

#### 3.2. Introducing a cutting plane

Creating contour plots across a plane is more convenient than isosurfaces. Cutting planes are the tools which can be used for this purpose, to create surfaces. This can be done by clicking on the *Slice* button in the Button Toolbars. A cutting plate can be manipulated and repositioned. In a similar way, the contour lines can also be derived out of planes.

By default ParaView triangulate the cells and shows them as triangles. For disabling this uncheck “triangulate the slice” option in the Properties panel of the slice.

#### 3.3. Streamlines

To create tracer lines, click on the *Stream Tracer* button in the Button Toolbars. Tracer points can be along a line or points, and this can be chosen in the *Seed Type* drop-down menu in the *Seeds* section of the Properties panel. Usually, some trial and error is needed for achieving the desired streamlines. The length of steps tracer takes can be changed in the *Streamline Parameters* section of the Properties panel. A smaller length increases calculation time but increases smoothness. For having high quality images *Tubes* filter can be used after tracer lines have been created. There are different types of tubes, not only cylindrical.

#### 3.4. Vector plots

The *Glyph* filter is used for creating vector plots. *Scale Mode* menu in the properties panel is used for:

- Setting the length of a vector, weather to be proportional to vector magnitude or not, all with the same length (*Vector*).
- Controlling the base length of the glyphs (set *Scale Factor*).

### 4. Data Analysis

#### 4.1. Plot over time

This option is available by clicking the *Plot Selection Over Time* button in the Button Toolbars. This allows the data at one point to be plotted over the entire time range.

#### 4.2. Plot over line

This option allows the data points to be plotted along a line at a specific time step. Click on the *Plot Over Line* button. The Cartesian coordinates of the beginning and ending points of the line can be specified in the Properties panel. Several variables

can be plotted at the same time, to turn each variable on or off and to change its legend name, use the Series Parameters section in the Properties panel.

### **4.3. Integrate Variables**

The *Integrate Variables* option is selected from the Filters menu. This tool integrates point and cell data over lines and surfaces. It also computes length of lines, area of surface, or volume<sup>[4]</sup>. Different data types available are *Point Data*, *Cell Data*, or *Field Data*; this can be chosen in the *Field Association* section in the Properties panel.

## **5. Exporting Data**

### **5.1. Image Output**

For creating a screenshot of the graphs the easiest way is Save Screenshot from File menu. After selecting it in the opened window the picture resolution can be set, and by locking the aspect ratio, changing image resolution in one direction cause change in its resolution in the other direction respectively. For high quality images a resolution of more than 1000 pixels is a good choice.

### **5.2. Animation Output**

Some animations can be saved in ParaView by selecting the *Save animation* option in the File menu. The resolution and number of frames per time step can be specified. You can save your animation by assigning a name and choosing the file format. The most suitable file format is *.ogv*.

### **5.3. Data Output**

The field values of a chosen variable (e.g. temperature or pressure) can be exported into Excel using the Save Data option in the File menu. The precision of the writer can be chosen and there is an option to export data from all time steps.