Blood Flow Aneurysm Tutorial Dataset Exploration

From VisItusers.org

Contents

Simulation Data

This tutorial uses the **aneurysm** dataset -- available at: http://www.visitusers.org/index.php?title=Tutorial_Data

To begin:

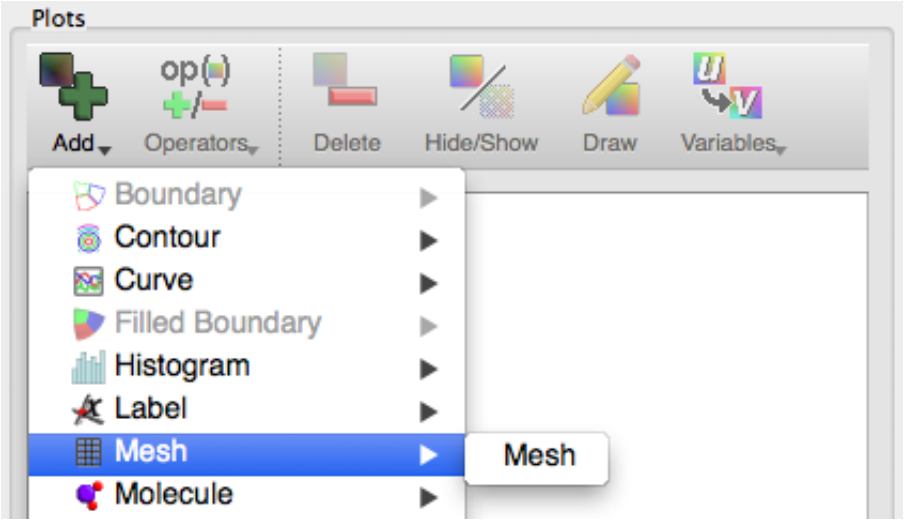
- Launch VisIt
- In VisIt's GUI, under the [Sources] section, click **Open**
- Navigate your file system to select the **aneurysm.visit** file.

Plotting Mesh Topology

First we will examine the finite element mesh used in blood flow simulation.

Create a Mesh Plot

- [Plot List] Add->Mesh->**Mesh**

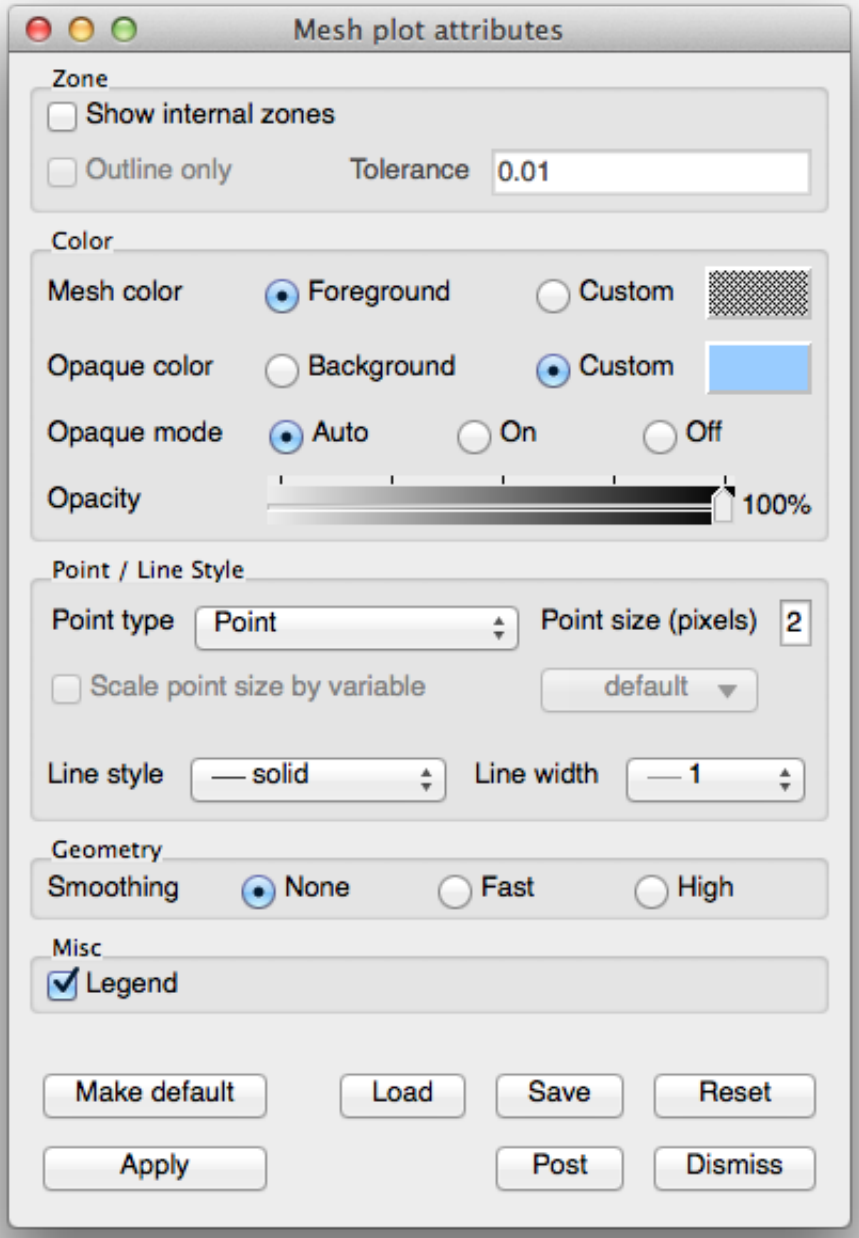


- [Plot list] Click **Draw**

After this, the mesh plot is rendered in VisIt's Viewer window. Modify the view by rotating and zooming in the viewer window.

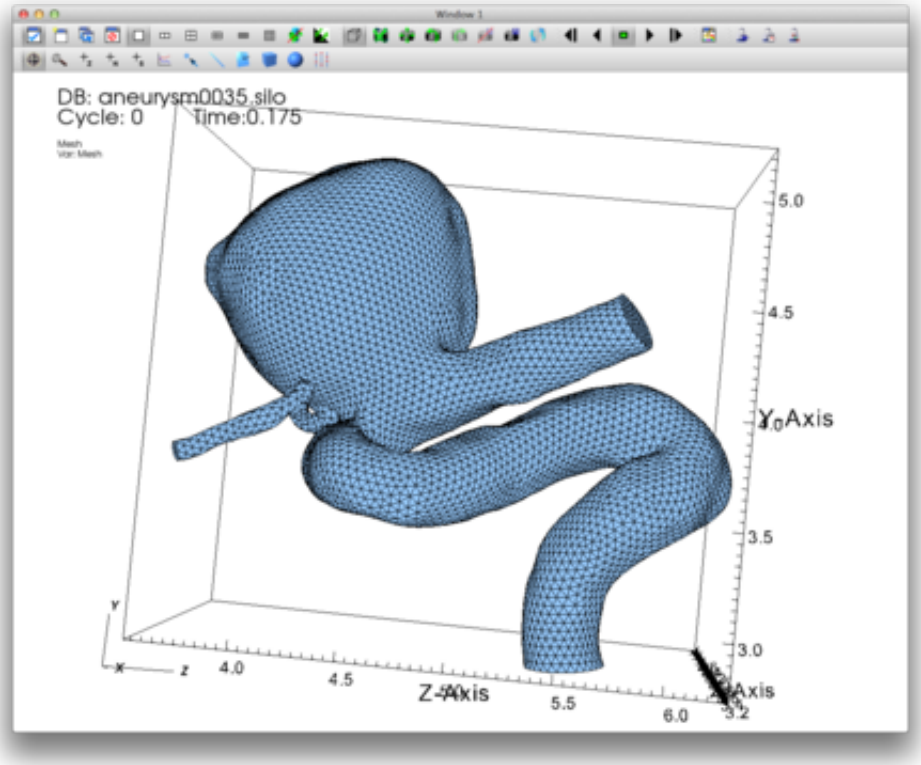
Modify the Mesh Plot Settings

- Expand the Mesh plot in the [Plot List] and double click to open the Mesh Plot Attributes Window.



- Experiment with settings for:
  - Mesh color
  - Opaque color
  - Opaque mode
  - Show internal zones

You will need to click **Apply** to commit the settings to your plot.



Query Mesh Properties

VisIt's Query interface provides several quantitative data summarization operations. We will use the query interface to learn some basic information about the simulation mesh.

- [Controls Menu]->Query
  - Select **NumZones** and click **Query**
    - This returns the number of elements in the mesh.
  - Select **NumNodes** and click **Query**
    - This returns the number of vertices in the mesh

Note: The terms **zones**, **elements**, and **cells** are overloaded in scientific visualization, as are the terms **nodes**, **points**, and **vertices**.

Exercises

- What type of finite element was used to construct the mesh?
- How many elements are used to construct the mesh?
- How many vertices are used to construct the mesh?
- On average, how many vertices are shared per element?

Examining Scalar Fields

In addition to the mesh topology, this dataset provide two mesh fields:

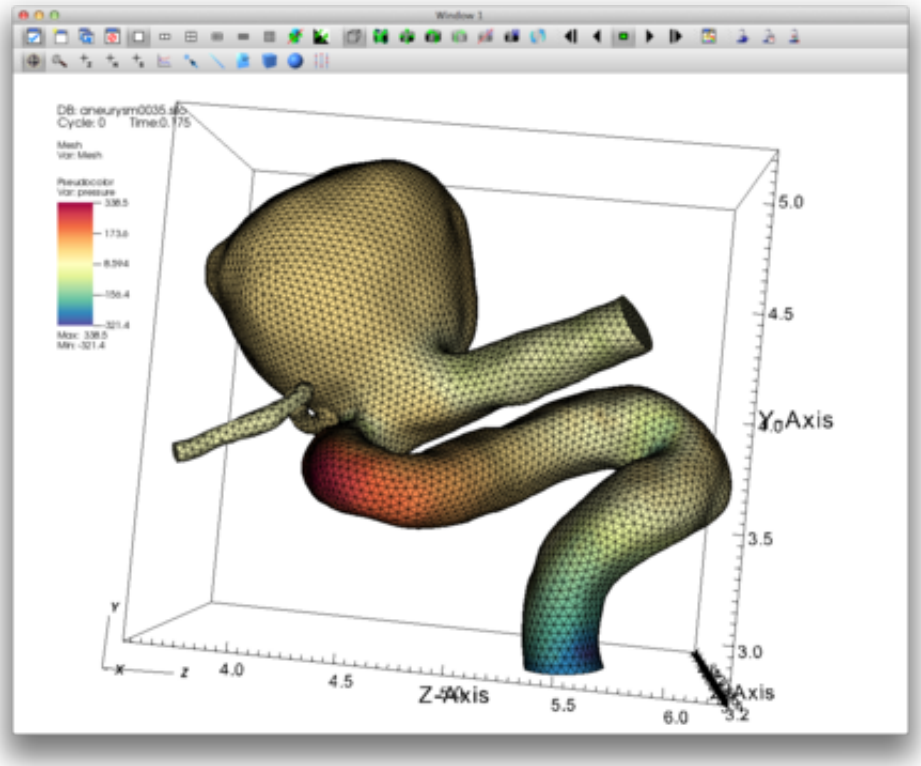
- A scalar field **pressure**, associated with the mesh vertices.
- A vector field **velocity**, associated with the mesh vertices.

VisIt automatically defines an expression that allows us to use the magnitude of the **velocity** vector field as a scalar field on the mesh. The result of the expression is a new field named **velocity_magnitude** .

We will use **Pseudocolor** Plots to examine the **pressure** and **velocity_magnitude** fields.

- [Plot List] Add->Pseudocolor->**Pressure**
- Expand the **Pseudocolor** plot and double click to bring up the **Pseudocolor Plot Attributes Window**.
- Change the color table to **Spectral** and check the **Invert** button
- Click **Apply**
- [Plot List] Click **Draw**
- [Time Slider] Click **Play**

You will see the pressure field animate on the exterior of the mesh as the simulation evolves.



- Experiment with:
  - Setting the **Pseudocolor** plot limits
  - Hiding and showing the **Mesh** plot

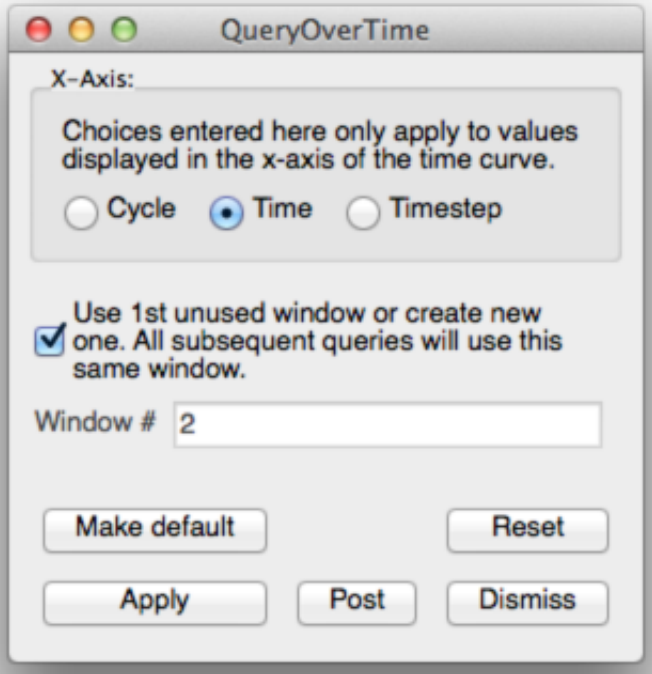
When you are done experimenting, stop animating over time steps using the [Time Slider] **Stop** button.

Query the Maximum Pressure Over Time

We can use the **pressure** field to extract the heart beat signal. We want to find the maximum pressure value across the mesh elements at each time step of our dataset. VisIt provides a **Query over time** mechanism that allows us to extract this data.

First, we need to set our query options to use **time** as the independent variable for our query.

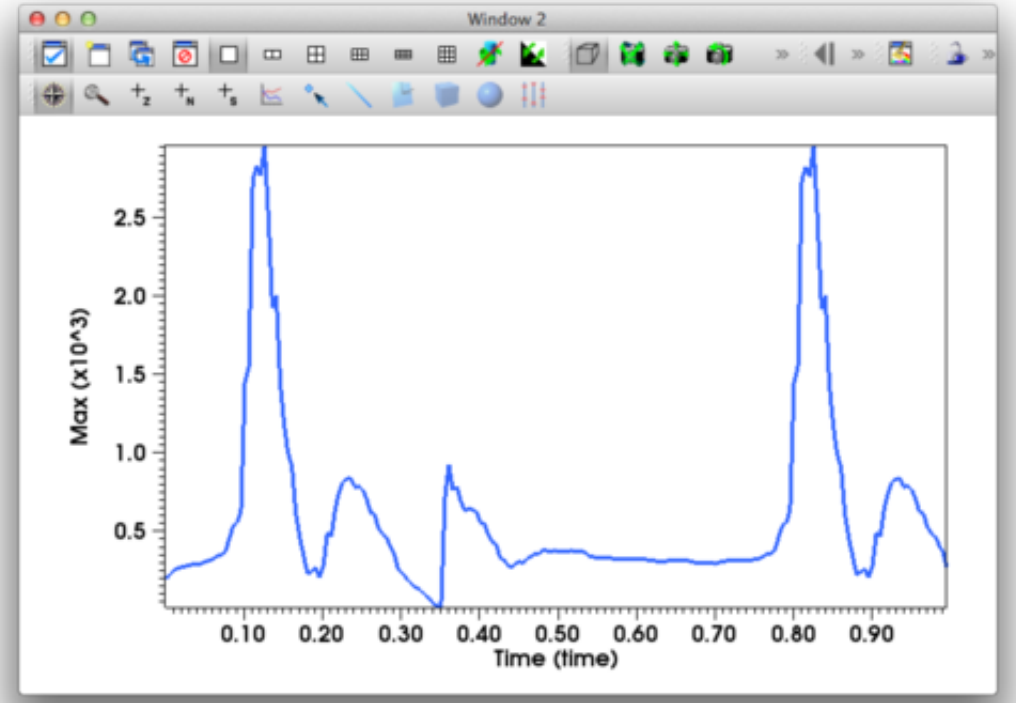
- [Controls Menu]->Query over time options
  - Select **Time**
  - Click **Apply** and dismiss the window



Now we can execute the **Max** query on all of our time steps and collect the results into a curve.

- [Plot list] Click to make sure your Pseudocolor plot is active
- [Controls Menu]->Queries
  - Select **Max**
  - Check **Do Time Query**
  - Click **Query**

This will process the simulation output files and create a new window with a curve that contains the maximum pressure value at each time.



Exercises

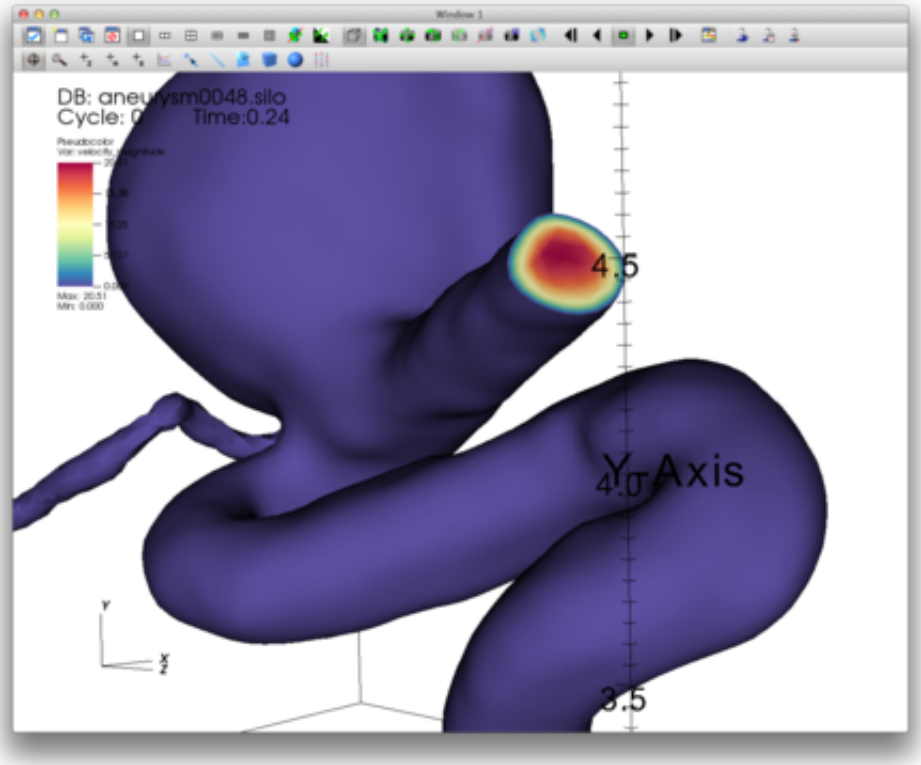
- How many heart beats does this dataset cover?
- Estimate how many beats per minute the simulated heart beating.

Contours and Sub-volumes of High Velocity

Examining the Velocity Magnitude

Next we create a **Pseudocolor** plot to look at the magnitude of the **velocity** vector field.

- [Plot List] Select and Delete your current plots
- Add a **Pseudocolor** Plot of **velocity_magnitude**
  - [Plot List] Add->Pseudocolor->**velocity_magnitude**
  - Open **Pseudocolor Plot Attributes Window** and set the color table options as before.
- [Plot List] Click **Draw**

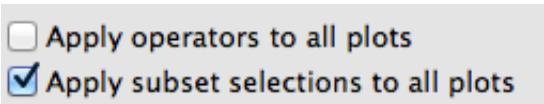


Notice that the velocity at the surface of the mesh is zero. To get a better understanding of the flow inside the mesh, we will use operators to extract regions of high blood flow.

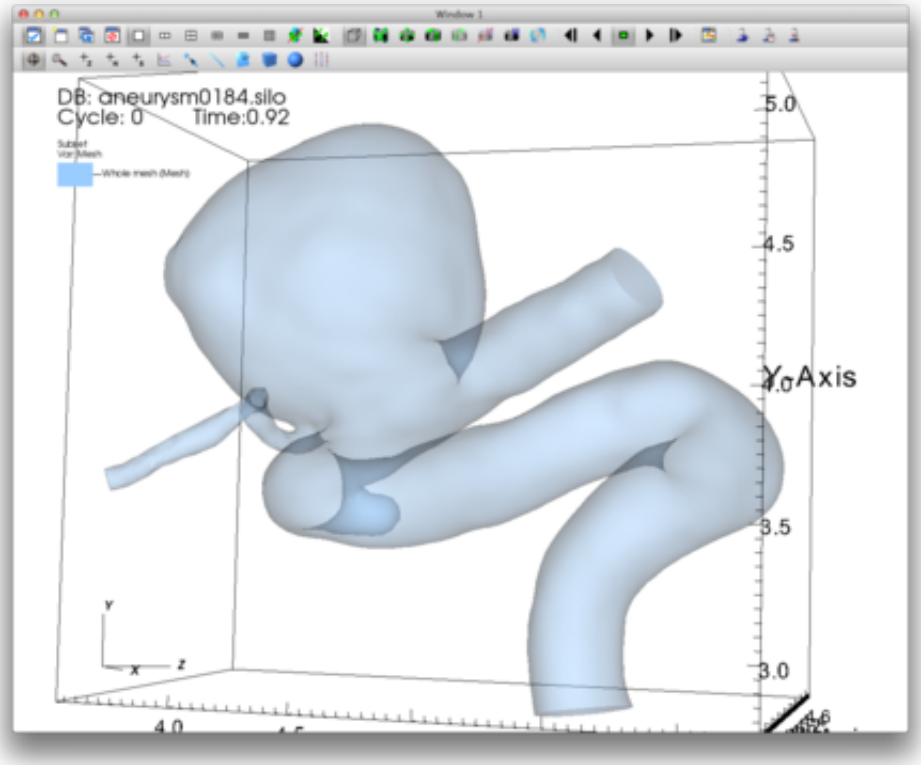
Creating a Semi-Transparent Exterior Mesh Plot

When looking at features inside the mesh, it helps to have a partially transparent view of the whole mesh boundary for reference. We will add a Subset plot to create this view of the mesh boundary.

- [Plot List] Uncheck **Apply operators to all plots**



- [Plot List] Add->Subset->**Mesh**
- Open the **Subset Plot Attributes Window**
  - Change the color to **Light Blue**
  - Set the **Opacity** slider to **25%**
  - Click **Apply**
- [Plot List] Click **Draw**

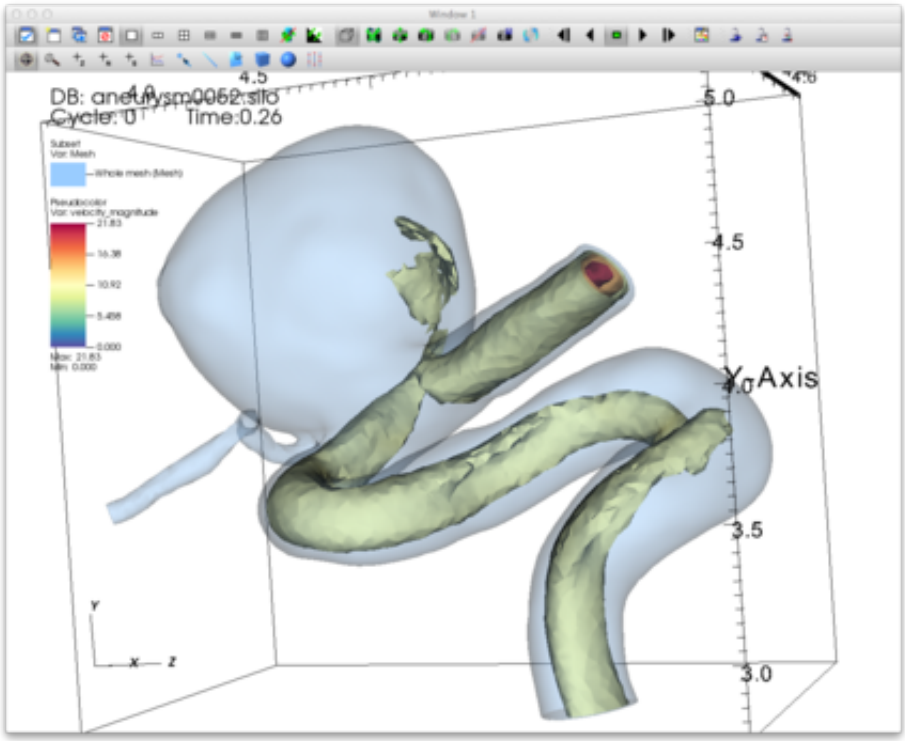


Contours of High Velocity

Now we will extract contour surfaces at high velocity values using the Isosurface Operator.

- [Plot list] Click to select your **Pseudocolor** Plot
- Add an **Isosurface** Operator
  - [Plot List] Operators->Slicing->**Isosurface**
- Open the **Isosurface Operator Attributes Window**
  - Set Select by to **Value**, and use **10 15 20**
  - Click **Apply** and dismiss the window.
- Click **Draw** and use the **Play** button to animate

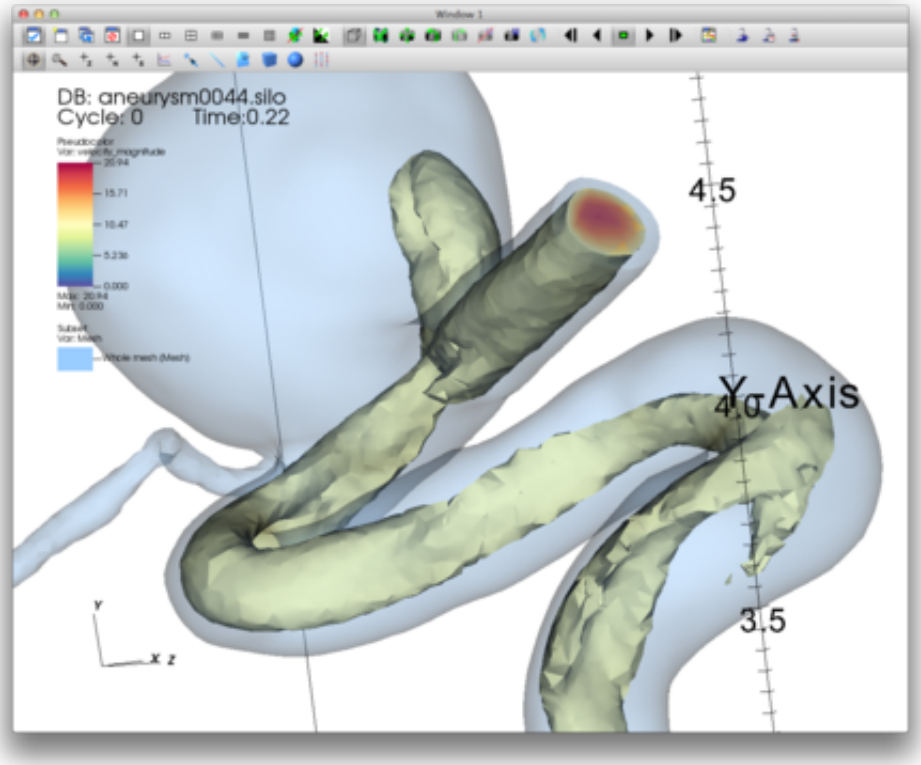
You will see the contour surfaces extracted from the **velocity_magnitude** field animate as the simulation evolves.



Sub-volumes of High Velocity

As an alternative to contours, we can also extract the sub-volume between two scalar values using the Isovolume Operator.

- [Time Slider] Click **Stop**
- Remove the **Isosurface** Operator
- Add an Isovolume Operator
  - [Plot List] Operators->Selection->'*Isovolume*
- Open the **Isovolume Operator Attributes Window**
  - Set the **Lower bound** to **10** and the **Upper Bound** to **20**
  - Click **Apply** and dismiss the window.
- Click **Draw** and use the **Play** button to animate



Retrieved from "http://visitusers.org/index.php?title=Blood_Flow_Aneurysm_Tutorial_Dataset_Exploration"

- This page was last modified 10:34, 4 November 2014.