

## How to Visualize Our Results in Paraview


### Getting Started

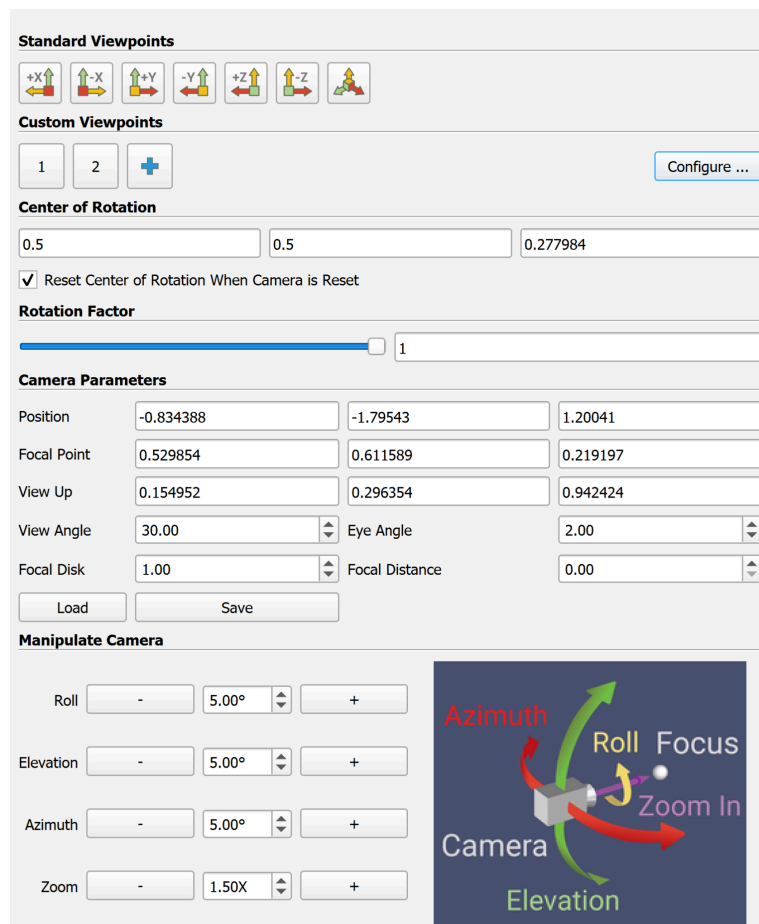
1. Open paraview
2. Click File - Open - VTK files you wish to open

### For 2D Plot of the Mesh and Solution

1. Click Apply in the Properties menu on the left hand side of the window
2. In the third drop down menu change Surface to Surface With Edges to see the mesh
3. To add a title : Filters - Search - Text - Add your Title - Apply

### For a 3D Surface Plot of the Mesh and Solution

1. Create the 2D plot of the solution and mesh
2. Filters - Search - Warp By Scalar - Apply
3. In the Properties menu on the left hand side of the window, turn on Data Axes Grid
4. In the third drop down menu change Surface to Surface With Edges to see the mesh on the surface plot
5. Click the  to change the perspective for viewing the surface. My saved viewpoint is



6. Add a title by adding another text filter (discussed above)

**Saving Your Progress:**

Save your plots: File - Save State - Pick File Path. Then you can open up the window as you left it at a different time in the future!