Visualization of Computational Fluid Dynamics OpenFOAM free surface flow simulations

Yun-Hang Cho, University of Sheffield, UK (IHPC, ASTAR)
yun-hang.cho@sheffield.ac.uk

Visualization is a vital part of the Computational Fluid Dynamics (CFD) workflow. A single image of the setup or results can give more information than a thousand lines of code. Open-source CFD software such as OpenFOAM are popular but does not come with a graphic user interface or visualization tools. Instead, general data visualization software such as ParaView are often used. This makes it difficult for users to learn how to display and interact with their work. For supercomputer users, additional complexities arise because the OpenFOAM is compiled with custom settings to improve performance and thus data in ParaView may not load/render at all.

Step 1. Loading the data

Whilst OpenFOAM contains parallelization functionality, hosting on supercomputing platforms require additional considerations.

For example, OpenFOAM uses 32-bit labels by default, however 64-bit labels are used on the NSCC Aspire-1 supercomputer.

Hence, one of the most common issues after downloading the data on a local visualization machine is that the ParaView software will assume the data contains 32-bit labels and crash with this error:

"...Expected punctuation token ')', found �)..."

In ParaView, advanced properties, the label size must be changed to 64-bit for the program to understand the data.

Step 2. Drawing attention to particular features

Once results data is loaded into ParaView, to show the computational domain, different faces may need different transparency.

For example, in Figure 1, the computational domains from two simulations can be overlaid and compared together by using different transparency. This is combined with an .STL file of the bed geometry through the **"pipeline" feature.**

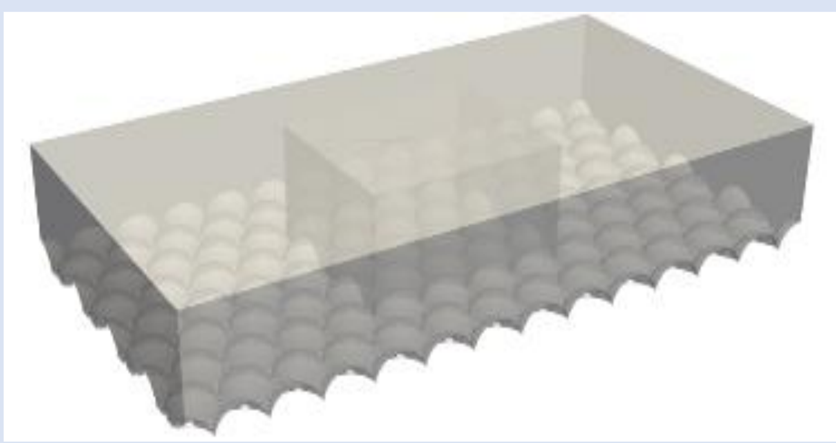

Figure 1. A larger domain is used to run Reynolds Averaged Navier-Stokes (RANS) simulation. A smaller domain is used to run an intensive Large Eddy Simulation (LES) simulation.

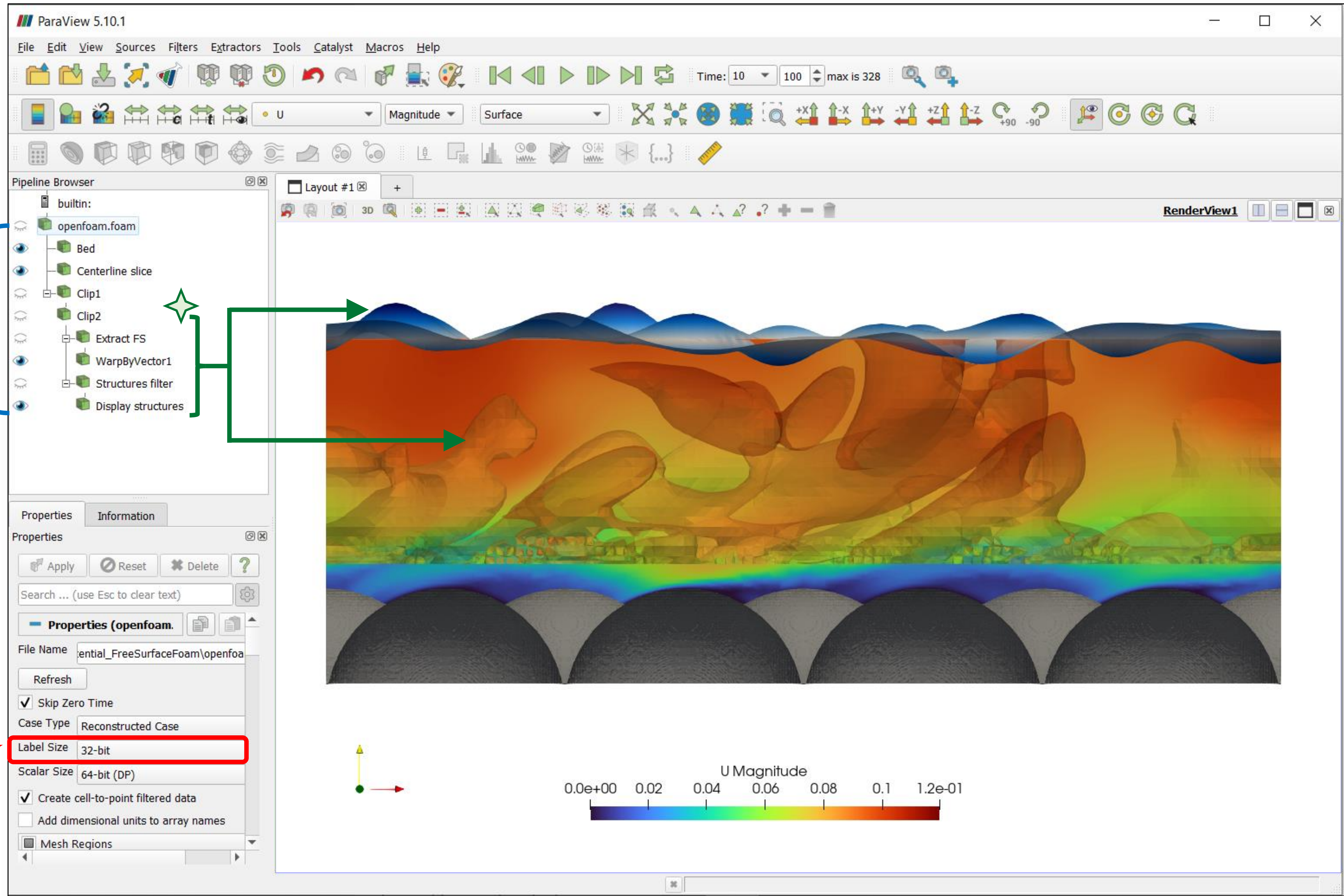

Figure 2. ParaView visualization of drainage open channel flow simulation over a bed with spherical roughness.

Step 3. Processing data visually to make new discoveries

Clever use of **"filters"** can even enable new scientific discoveries.

- In Figure 2, the "Gradient of Unstructured Data" filter calculates the vorticity showing how turbulent vortices move within the flow.
- The "Warp by Vector" filter uses the velocity near the water surface to model/simulate the shape of the water water surface.
- Together, we can see that the turbulent vortices hit the water surface and cause it to deform more than usual.

National Supercomputing Centre Singapore

The University Of Sheffield.

Agency for Science, Technology and Research Singapore

References: Cho Y-H, Dao MH and Nichols A (2022) Computational fluid dynamics simulation of rough bed open channels using openFOAM. *Front. Environ. Sci.* 10:981680. doi: 10.3389/fenvs.2022.981680

Acknowledgements
Sincere thanks to the NSCC for hosting the simulation and to the ASTAR for partly funding this work. Additional thanks to Dr Andrew Nichols and Dr Dao My Ha at the Institute of High-Performance Computing for their guidance and supervision.

SCA 2023 Supercomputing Asia
Gathering the **Best of HPC** in Asia

HPC Asia 2023

CoNGA