Visualization of Computational Fluid Dynamics OpenFOAM free surface flow simulations

Yun-Hang Cho
University of Sheffield
UK
yun-hang.cho@sheffield.ac.uk

1. INTRODUCTION

With climate change increasing flood risk, it is critical to improve our understanding of how waterway designs interact with the flow turbulence, pollutant transportation and overall flow carrying capacity. One common type of waterway used to transport ground water is the open channel. Computational Fluid Dynamics (CFD) simulation software such as OpenFOAM are a good tool to help simulate open channel flows but does not include a graphic user interface or any visualization tools. Whilst this gives the user more freedom in visualization, it also makes it more difficult for beginners to learn how to display and interact with their work. Visualization is a vital part of the CFD workflow as a single image of your setup or results can indicate a critical error that may not be clearly evident in a thousand line of code. The most used visualization tool with OpenFOAM is ParaView. For supercomputer users, it also presents additional challenges that are not normally covered in the online manuals [1]. This poster will presents some of the challenges facing OpenFOAM users with visualizing their work and the solution. The topics have been selected based on the likelihood of the problem arising as well as the difficulty in finding a suitable answer through online forums. For HPC Asia, the topics are also focused around the author's experience with deploying OpenFOAM simulations on supercomputing facilities such as the National Supercomputing Center's Aspire-1 supercomputer.

2. Challenge 1: Loading the data

Whilst OpenFOAM itself contains parallelization functionality, hosting OpenFOAM on supercomputing platforms require additional considerations and optimization. This results in settings which are no longer default. For example, OpenFOAM installs with 32-bit labels by default, however 64-bit label are used on the supercomputer. Hence, one of the most common issues when first loading OpenFOAM data on a local visualization machine is that the ParaView software will assume the data contains 32 bit labels and crash (it says: Expected punctuation token ')', found). An advanced setting in ParaView is required to change the reader to display the results (see Figure 1).
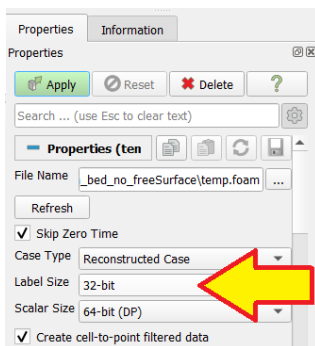

Figure 1. Change the label size to 64-bit to display results run using NSCC's Aspire-1 supercomputer.

3. Challenge 2: Visualizing the data

Once the data is loaded into ParaView, it maybe necessary to have different transparency for different faces of the computational domain. For example, in Figure 2, the computational domains of two simulations are displayed with together with different transparency in addition to an .STL file of the bed geometry. Clever use of the "pipeline" feature can allow different surfaces of the same simulation or different simulations to be altered so that both the turbulent coherent structures within an open channel flow and the free surface can be seen at the same time (Figure 3).
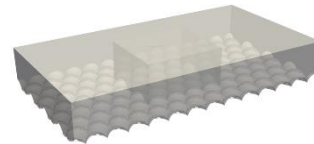

Figure 2. A larger domain is used to run a lighter simulation with Reynolds Averaged Navier-Stokes (RANS) turbulence modelling whilst a smaller domain is used to run a heavier Large Eddy Simulation (LES) [2].
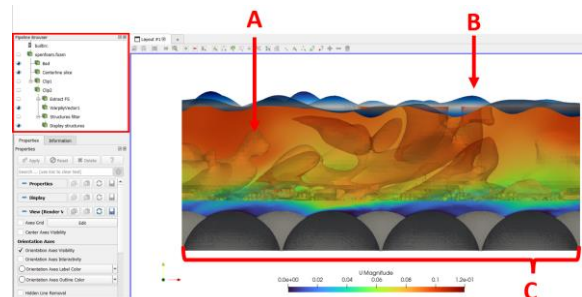

Figure 3. Visualization of flow over a rough bed of spherical caps (C). The free surface (B) with the "warp by vector" function with turbulent structures identified using Q-criteron (A). Colorbar represents the streamwise velocity.

To summarize, if used correctly, ParaView contains powerful tools for OpenFOAM flow visualization and animation.

ACKNOWLEDGMENTS

REFERENCES

[1] OpenFOAM online user manual, URL: openfoam.com/documentation/user-guide, accessed 06/0122.

[2] 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