Performance and Usage Improvements for Multiple OpenFOAM Customers: HPC and Cloud

Ning Li Sally Bridgwater Nick Dingle Jon Gibson (Numerical Algorithms Group)

Background

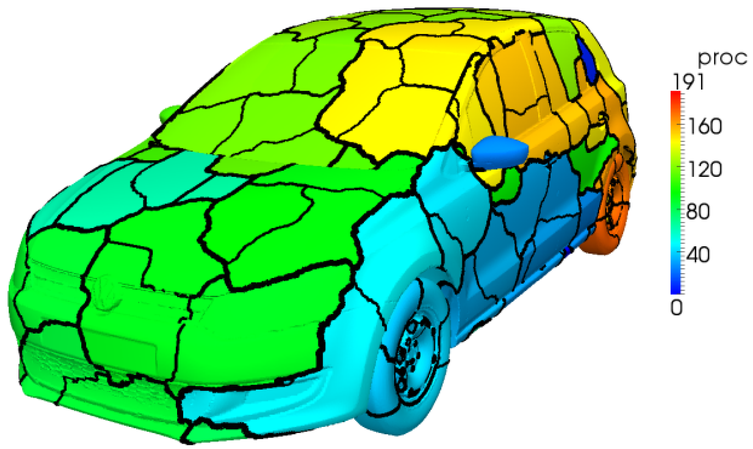
This poster describes work performed on OpenFOAM focussing on performance as well as OpenFOAM in the Cloud.

OpenFOAM is a C++ toolbox, parallelized with MPI, for the development of customized numerical solvers for the solution of continuum mechanics problems, including computational fluid dynamics.

Ease-of-Use in the Cloud for Volkswagen

NAG participated in a Fortissimo experiment alongside RWTH Aachen to help end users from Volkswagen move some of their OpenFOAM simulations onto cloud-based HPC platforms. NAG's main roles here included:

- Studying the overall architecture and software environment of the hosting clusters and supporting partners
- Streamlining the software deployment process for OpenFOAM for use without specialist technical knowledge by end users
- Performing typical Computational Science and Engineering (CSE) support work e.g., parallel scaling studies of the test cases
- Developing a cost analysis model to demostrate the benefit of using cloud-based solutions



Geometry of the Polo Mk V car body with mesh decomposition over 192 cores. NAG performed external aerodynamics simulations of this real car model on the ARCHER supercomputer to validate the software deployment and perform scaling studies.

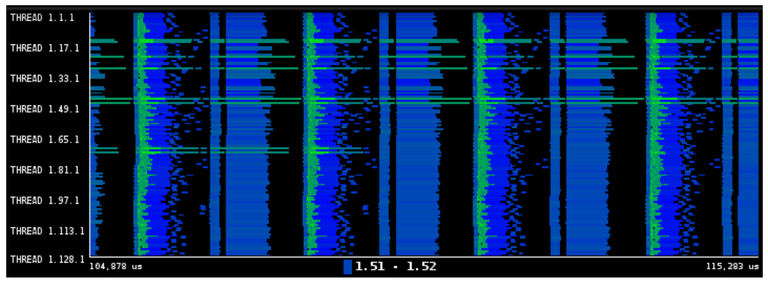
This work allowed Volkswagen to make well informed decisions on the best path forwards and to quickly get started using cloud-based solutions.

Performance Assessment of Microclimate Solver

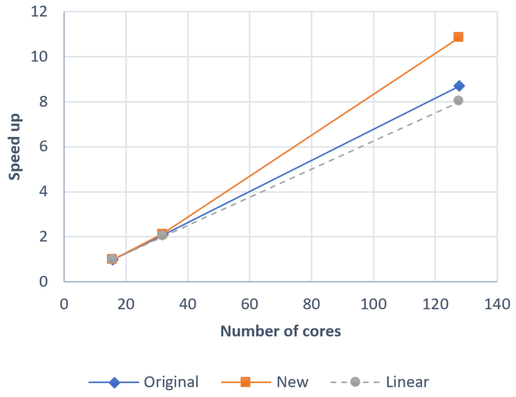
The performance of the uhiSolver (Urban Heat Island Solver), developed by Rheologic GmbH, was assessed. This is developed using the OpenFOAM framework to forecast local conditions in urban areas whilst including the cooling effects of plants and water surfaces due to evaporation.

Overall the application was found to scale well with super-linear scaling due to increased instructions per cycle (IPC) with core count.

Load imbalance was found to be causing inefficiency in the solver. We identified that this was because of higher cache miss rates on some processes which resulted in some slow processes holding up the others.



A recommendation was made was to improve the locality of data in order to reduce the cache misses. The CFD mesh used in the simulation gets decomposed into cells (domains). The new implementation now calls `renumberMesh[]` which renumbers the cell list during the initialization phase.



25% reduction in time-to-solution was achieved with less than one day of development by Rheologic.

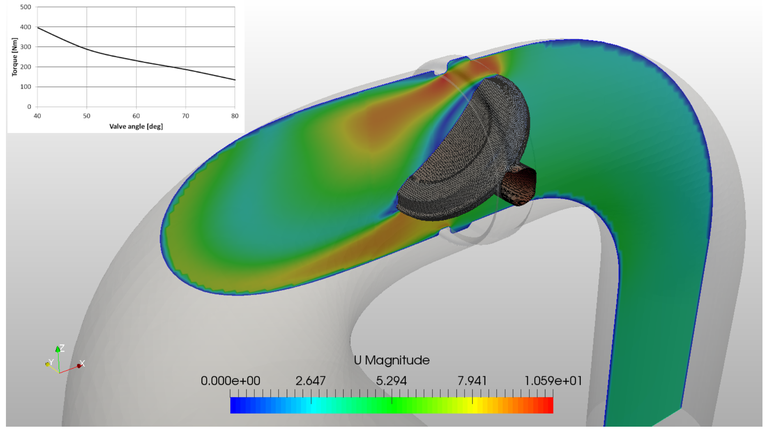
HELYX in the Cloud

NAG helped ISV partner ENGYS to implement a client-server component in HELYX to allow job set-up, execution, and pre- & post-processing on a remote cluster via a desktop environment.

HELYX is ENGYS's flagship CFD package, a much enhanced OpenFOAM with additional and often better solvers and utilities, together with a graphical user interface.

NAG's main roles included: automating the deployment of the server component of HELYX; establishing best practice to facilitate remote visualization of large distributed datasets. ENGYS then implemented the logic to support visualization within the GUI of HELYX.

This capability was then used by an end user, Dynaflow, for their oil and gas simulations of the properties of pipeline components.



Acknowledgements

The research leading to these results has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No. 676553.

The research leading to these results has received funding from the European Union Seventh Framework Programme [FP7/2007- 2013] under grant agreement no. 609029.

Performance Optimisation and Productivity

FORTISSIMO