OpenFOAM Performance Optimizations for Scalability

Pak Lui

HPC Advisory Council
350 Oakmead Parkway, Suite 100, Sunnyvale, California 94085, USA

OpenFOAM is an open source CFD simulation software suite which allows users to produce high-quality realistic simulation for visualization in many industries. It provides a range of solvers with a high degree of parallelism and scalability capabilities that enables it to take full advantage of multi-core HPC clusters. By carrying out the simulations on HPC systems in parallel in order to minimize the time needed to generate the simulation quickly and cost-effectively. To achieve good scalability performance on the HPC scientific applications sometimes involves good understanding of the workload though performing profile analysis, and comparing behaviors of using different hardware which pinpoint bottlenecks in different areas of the HPC cluster. In this topic, the HPC Advisory Council has performed a deep investigation and profiling for some of the OpenFOAM solvers to understand its performance and scaling capabilities using the high-speed InfiniBand networks and to explore potential optimizations. We will demonstrate various methods of profiling and analysis to determine the bottleneck, and the effectiveness of the tuning to improve on the application performance. This topic also presents the optimization techniques and networking profiling results to further understand the dependencies of some of the OpenFOAM solvers on the network and the MPI library, and options for optimizations using MPI offloads to particular OpenFOAM solvers.