Large Scale Transient CFD Simulations for Buildings using OpenFOAM on a World's Top-class Supercomputer

Pham Van Phuc, Shuichi Chiba, Kazuo Minami

Shimizu Corporation, 4-17, Etchujima 3-Chome, Koto-ku, Tokyo, Japan, 135-8530, p_phuc@shimz.co.jp
Fujitsu Limited, 140, Miyamoto, Numazu, Shizuoka, Japan, 410-0396, shuc@jp.fujitsu.com
RIKEN, 7-1-26 Minatojima-minami-machi, Chuo-ku, Kobe, Hyogo 650-0047, Japan, minami_kaz@riken.jp

Computational fluid dynamics (CFD) simulation is understood as a powerful tool which will able us to study, assess the fluid flow phenomena and to improve their effects. It is expected to be used widely in building engineering applications, nowadays. OpenFOAM is well known as a robust CFD platform implemented using the C++ programming language, including a number of feasible solvers and useful pre- and post-processing tools for multi-physics applications. It expedites transparency in numerical modelling and productivity for customizations to the CFD applications.

Because of the extensive range of features, OpenFOAM has become an attractive subject of our studies in different aspects, not only the accuracy and utility of CFD but also the capability of a large scale CFD simulation, or advanced compiler design and hardware-software system co-design with a C++ software, etc. We conducted several studies for building applications related to wind flow field as well as free surface flow field, and confirmed the accuracy of OpenFOAM in comparison with experiment and field measurement results. Therefore, OpenFOAM has begun to permeate through our implementation design phase. Some efforts have also been made for OpenFOAM with High Performance Computing Supercomputers in the first stage. Under some specific modifications and tuning for OpenFOAM and its pre- and post-processing tools, we have succeeded in generating the model and doing a large scale transient CFD simulation up to **100 billion cell meshes** and achieved a high performance for **100 thousand MPI** parallels using **the K Computer (the world's fastest supercomputer by TOP500 in 2011)**.

In the presentation, implementations of OpenFOAM for building engineering applications (wind environment of urban city, wind or tsunami loads acting on buildings, flow field controls including verification, validation, advanced turbulence models and practical results, etc.) are comprehensively introduced. Large scale transient CFD simulation using **the K Computer** with **sophisticated LES results** for prediction of the critical wind pressure acting on buildings is the main topic for discussion. Challenging aspects, solutions and benchmark results are also focused to clarify the significant problems, potential abilities and capabilities of OpenFOAM in the simulation up to the **hundred billion cell meshes** or/and **the hundred thousand MPI parallels** on the World's Top-class Supercomputers. Finally, our recent activities are introduced and concluding remarks are given. We look forward to the continuing growth of OpenFOAM and also encourage fruitful collaboration with its strong global communities in near future.