11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

PRACTICAL APPLICATIONS FOR THE COMPUTATIONAL VEHICE AERODYNAMICS ON THE MASSIVELY PARALLEL SUPERCOMPUTER: PART 1, FRAMEWORK FOR THE FULLY UNSTRUCTURED FINITE VOLUME CELLS

Andrew H. Kerr[1], Keiji Onishi[2] and Makoto Tsubokura[34]

[1] Hokkaido University, Faculty of Engineering, Sapporo, Japan, ahkerr@ec.hokudai.ac.jp
[2] RIKEN Advanced Institute for Computational Science, Kobe, Japan, keiji.onishi@riken.jp
[3] Hokkaido University, Faculty of Engineering, Sapporo, Japan, mtsubo@eng.hokudai.ac.jp
[4] RIKEN Advanced Institute for Computational Science, Kobe, Japan, mtsubo@eng.hokudai.ac.jp

Key Words: *Aerodynamics, Vehicle, Large-Eddy Simulation, K Computer, Finite Volume Method*

A simulation framework for vehicle aerodynamics using up to 10 billion fully-unstructured numerical elements has been developed on the world fastest class supercomputer called the “K computer”. The K computer is installed at the RIKEN Advanced Institute for Computational Science in Kobe, Japan, consisting of over 80,000 processors of a distributed memory architecture with its peak performance of about 10 PFLOPS, and ranked as the world’s fourth-fastest computer as of June, 2013.

The simulation software “FrontFlow/red-Aero”, developed by our group, was fully optimized on the K computer to utilize up to 10,000 processors with tens of thousands of cores. A hybrid parallelization method using MPI (Message Passing Interface) among multiple processors and OpenMP among cores inside the processor was adopted. The software was optimized for unsteady aerodynamic simulation, thus wall-modeled LES (Large-Eddy Simulation) was adopted for the turbulence simulation. To solve the pressure field and its coupling with the velocity field, an incompressible flow algorithm based on SMAC (simplified marker and cell) was originally adopted. However it was recognized in this study that such a massively large number of elements requires an excessive amount of iterations for convergence of the pressure Poisson solver. Thus we have newly adopted a weakly compressible algorithm to reduce the computational load required to solve the pressure field.

On the other hand, the challenging issue to realize a massively parallel simulation in terms of vehicle aerodynamics is not only to tune the simulation software but also to generate the billions of fully-unstructured numerical elements. Thus we have developed an automated mesh refining system. In the system, users only generate unstructured grids in the order of tens of millions of elements by hand using commercial preprocessing software. Then the original elements are refined in the preconditioning process using the K computer, billions of elements are automatically generated and the simulation is conducted. In the same way, if a user requires, the results of the billions of elements are mapped on the original coarse grid during post processing and the user can transport the data from the K computer without any stress. Then the results can be visualized using conventional software. That is to say, the user can conduct the massively parallel simulation of billions of elements, without the consciousness of doing so, in a conventional CAE environment.

The simulation framework was validated on vehicles with different rear slant angles, and dependence of the aerodynamic forces on the unstructured-grid resolution as well as the grid topology near the surface was discussed. In this study, two topologies of the grid near the wall were compared; one is a full tetrahedral grid (an original tens of millions of elements and then refined billions of elements) and the other is the hybrid tetra/prism grid (original tens of millions of elements). The advantage of the full tetra mesh is its ease of mesh generation, while the troublesome prism meshes on the surface are always necessary in the conventional CAE environment under the restriction of the computational resources with the total element number of less than one hundred million. In this study we have confirmed that the simulation using the multi-billion element, auto-refined tetra mesh on the K computer shows very good agreement with wind-tunnel measurements within a small percentage of error. Finally some issues with regard to surface modification from so-called dirty CAD data for fully unstructured grids are discussed.