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 VEHICLE AERODYNAMICS ON THE MASSIVELY PARALLEL SUPERCOMPUTER: PART 2, HIERARCHICAL CARTESIAN GRID APPROACH UTILIZING DIRTY CAD DATA

Keiji Onishi[1], Makoto Tsubokura[2], Shigeru Obayashi[3] and Kazuhiro Nakahashi[4]

[1] RIKEN Advanced Institute for Computational Science, Kobe, Japan, keiji.onishi@riken.jp
[2] RIKEN Advanced Institute for Computational Science, Kobe, Japan, mtsubo@eng.hokudai.ac.jp
[3] Tohoku University, Institute of Fluid Science, Sendai, Japan, obayashi@ifs.tohoku.ac.jp
[4] Japan Aerospace Exploration Agency (JAXA), Tokyo, Japan, nakahashi.kazuhiro@jaxa.jp

Key Words: *Aerodynamics, Vehicle, Dirty CAD, Building Cube, Immersed Boundary, Large-Eddy Simulation, K Computer, Cartesian grid*

The objective of this research is to propose the methodology to enable the user to reduce manual work in preparing geometry for simulation of vehicle aerodynamics when working with ‘dirty’ computer-aided-design (CAD) data. When conducting simulations based on computational fluid dynamics (CFD) in this field, there remains a major problem that the user must carry out much work in generating a high-quality grid, although the user uses an unstructured grid or Cartesian grid. This heavy workload of geometry preparation strongly hinders the broadening of applications in the automotive CFD field, and as such, it is an important issue to resolve. Recent CFD researches show the Cartesian grid approach is reinstated, especially the immersed boundary method (IBM). It has an advantage for the treatment of complex geometries, but the challenging is a wall treatment. And it has a limitation of the requirement for geometry which cannot have a gap, overlap, and thin wall. Hence, a new methodology that enables a solver to treat such geometries directly was developed with modified IBM approach, and the result of having resolved near the wall well using high performance computing (HPC) is discussed in this study.
The validation case of flow around a sphere with high Reynolds number is presented, adopting the close conditions to the vehicle aerodynamic analysis. The large eddy simulation (LES) results with the coarse grid shows a large discrepancy with past studies that shows the laminarized result on pressure coefficient distribution and the over predicted value of drag coefficient (Cd). The huge calculation using 23 billion cells shows an improvement for Cd prediction on the LES configuration.
A vehicle aerodynamics CFD analysis is conducted using the dirty CAD data using about 20 billion cells. The heat exchanger model using an additional forcing term and the closing volume model to prevent flow penetrating into the vehicle inside is considered. The reasonable flow results such as a flow inside engine bay and shear layer around front thin spoiler are observed. The calculation can be started within few hours from when the CAD data is ready. This has a big impact on the automotive design field because they have to spend more time, almost 1 week, in present, from when they designed a new production model on their CAD system. According as the computational architectures development in near future, the proposed method has a possibility to become an innovative scheme in vehicle aerodynamics CFD to change the design process by the advantage shown in this research.