Proceedings of FEDSM2007
5th Joint ASME/JSME Fluids Engineering Conference
July 30-August 2, 2007 San Diego, California USA

FEDSM2007-37456

CURRENT STATUS ON HIGH PERFORMANCE COMPUTING FOR VEHICLE AERODYNAMICS USING LARGE EDDY SIMULATION

Makoto Tsubokura[*]
Department of Mechanical
Engineering and Intelligent Systems,
The University of Electro-
Communications,
Chofu-shi, Tokyo, Japan
Email: tsubo@mce.uec.ac.jp

Takuji Nakashima
Department of Social and
Environmental Engineering,
Graduate School of Engineering,
Hiroshima University, Japan
Email: nakashima@hiroshima-u.ac.jp

Nobuyuki Oshima
Division of Mechanical and Space Engineering,
Graduate School of Engineering,
Hokkaido University, Japan
Email: oshima@eng.hokudai.ac.jp

Kozo Kitoh
Kozo Kitoh Technology, Inc, Japan
Email: kozo-kitoh@cam.hi-ho.ne.jp

Huilai Zhang
Advance Soft, Japan
Email: zhang@advancesoft.jp

Keiji Onishi
Advance Soft, Japan
Email: onishi@advancesoft.jp

Toshio Kobayashi
Japan Automobile Research Institute, Japan
Email: kobaya@jari.or.jp

ABSTRACT

The world's largest class unsteady turbulence simulations of flow around vehicles were conducted using Large Eddy Simulation (LES) on the Earth Simulator in Japan. The main objective of our study is to investigate the validity of LES, as an alternative to a conventional wind tunnel measurement or the Reynolds Averaged Navier-Stokes method, for the assessment of vehicle aerodynamics.

INTRODUCTION

Computational Fluid Dynamics (CFD) is going to be a powerful tool for the vehicle aerodynamics from the viewpoint of its enormous amount of information as well as high economic efficiency. However, the Reynolds Averaged Navier-Stokes (RANS) method commonly used for vehicle aerodynamics has two fun-

---

[*] Address all correspondence to this author.

damental problems: one is it is strongly depending on the turbulence model, and the other is it only predicts the averaged flow characteristics. Thus RANS only plays a supplementary role of a wind tunnel test at the moment.

Recently, greater attention is paid to unsteady aerodynamic force generated from sudden steering action, overtaking, or cross wind, all of which are difficult to estimate not only by RANS method but also by a wind tunnel test, and an alternative method to the conventional manners is strongly desired. Large Eddy Simulation (LES) will be an encouraging solution to the problem, because it can reproduce unsteady turbulence characteristics with high accuracy, but in turn it requires excessively large computational resources. Consequently only few attempts have been made so far to apply LES to the assessment of vehicle aerodynamics. The objective of this study is to show the validity of LES using high-performance computing technique ((HPC-LES)) for the assessment of vehicle aerodynamics, as a computer aided engineering (CAE) tool for the next generation.

For this purpose, the unstructured Finite Volume software 'FrontFlow/red' developed at the Univ. of Tokyo was intensively optimized for the execution on the Earth Simulator, which made it possible to conduct the unsteady turbulence simulations around vehicles using at most 120 million elements on 800 parallel processors.

We first applied the method to the flow around the aerodynamic model ASMO(Aerodynamisches Studien Modell) to show the validity of the method by comparing the results with existing wind-tunnel data. Then it was successfully applied to a racing motorcycle, a commercial sedan, and a formula car.

NUMERICAL METHODS

Governing Equations

The governing equations for LES adopted in this study are the spatially filtered continuity and Navier-Stokes equations as indicated below:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} 2(\nu + \nu_{sgs}) \bar{S}_{ij}, \tag{2}$$

$$\bar{S}_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right), \tag{3}$$

$$P = (\bar{p}/\rho) + \frac{1}{3}(\overline{u_k u_k} - \bar{u}_k \bar{u}_k), \tag{4}$$

in which $\bar{u}_i$,$\bar{p}$,ν, and ρ are the spatially filtered-velocity, pressure, kinetic viscosity, and density, respectively. The last term on the right of eq. (2) is the molecular and the subgrid-scale (SGS) eddy viscosity term. The SGS eddy viscosity is modeled as,

$$\nu_{sgs} = (C_s f_s \Delta)^2 \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}, \qquad \Delta = (V_{element})^{1/3} \tag{5}$$

The model coefficient in eq. (5) is given as Cs=0.15 in this study. The Van-Driest type damping function $f_s = 1 - \exp(-y^+/25)$ is adopted. The SGS model is called Smagorinsky 's eddy viscosity model.

Discretization

The governing equations are discretized based on the vertex-centered unstructured finite volume method. The central finite difference scheme with the second order accuracy is adopted for spatial discretization except for the convective term in which about 10% (depending on the cases) of the first order upwind scheme is blended for the numerical stability, while the second order Adams-Bashforth scheme is adopted for time marching based on the SMAC method.

Numerical Conditions

Boundary conditions for the computation are applied based on the real wind tunnel test. Uniform velocity condition is imposed at the inlet, while free-outlet condition is given at the exit of the computational domain. For the accuracy of the outlet condition, we attached an additional layer consisting of prism elements at the exit of the domain to align the grid lines with the mean velocity direction. On the surfaces of the car body, a solid wall condition is adopted. The typical wall distance of the first nearest gird point measured from the numerical results are about 120 for ASMO and 40 $\sim$ 80 for the formula car in wall unit, which are located within the logarithmic layer of the mean velocity profile. The assumed log-law profile is directly applied to the instantaneous velocity field to estimate the surface friction. On the floor, free slip condition is applied from the inlet to just ahead of the car to prevent the boundary layer from developing. In the real wind tunnel test, a boundary suction system is mounted on the floor ahead of the car. On the floor behind the car toward the exit, the solid wall condition is given. The free-slip wall condition is imposed also on the ceiling and sidewall of the numerical domain.

The hardware on which we conducted our CFD is the Earth Simulator (ES), which has been operated since 2002 at the Japan Agency for Marine-Earth Science and Technology. The ES was developed under the national project by Japanese governmental agencies. The ES is a highly parallel vector supercomputer system of the distributed memory, and consisted of 640 processor nodes (PN). Each PN consists of 8 vector-type CPU (peak performance is 8 Gflops/CPU) with a 16 Gbytes memory system, consequently the total peak performance is 40 Tflops with maximum memory of 10 Tbytes. From 2002 to 2004, the ES has been

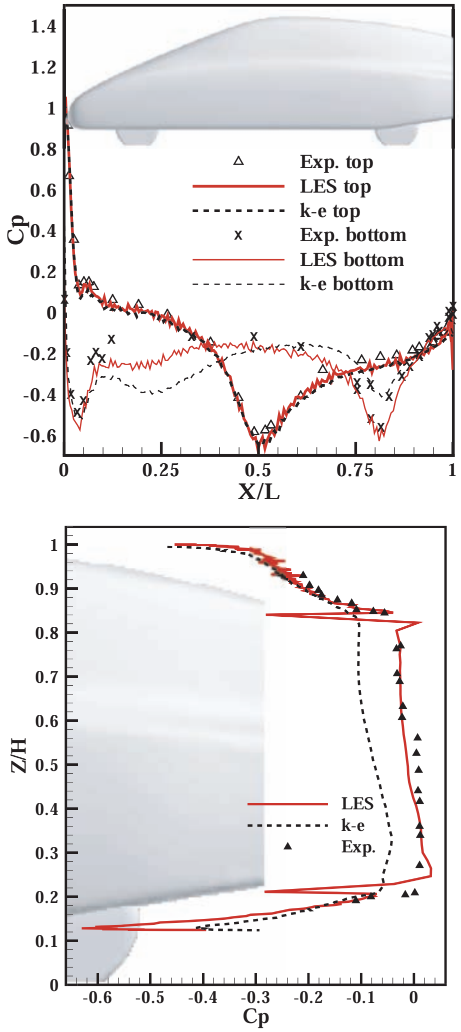

Figure 1. SURFACE MEAN PRESSURE DISTRIBUTIONS ON THE ASMO MODEL

recognized as the world fastest supercomputer in the world, and still now, as a practical tool for real engineering problems, the ES is one of the fastest in the world.

The computational code adopted is the FrontFlow/Red having been developed under the project of 'Frontier Simulation Software for Industrial Science' organized by Professor Toshio Kobayashi at the University of Tokyo at the time. The project started in 2002 as an IT-program research project sponsored by the Ministry of Education, Culture, Sport, Science and Technology, and was successfully concluded in 2005. The code was intensively optimized by ourselves on the ES under the successive renovated IT project 'Revolutionary Simulation Software (RSS21)' organized by Professor Chisachi Kato, and high vectorization and parallelization of more than 96% and 99% respectively were achieved in the simulations on 100PNs/800CPUs, which made it possible to complete the large-scale computations (at most 120 million meshes with more than 500 GB memory) including the calculation of time-averaging turbulence statistics in about 120 hours.

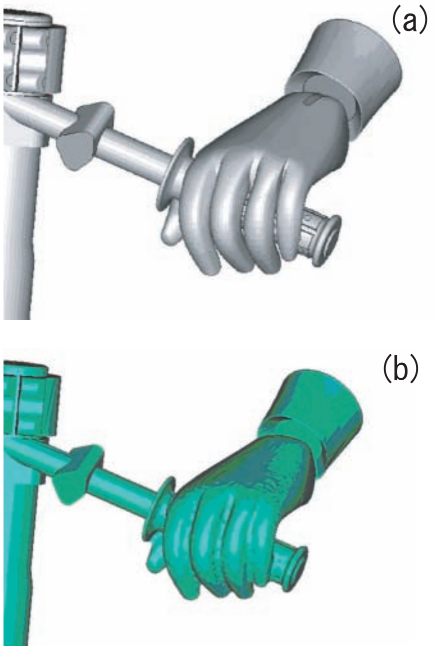

Figure 2. SURFACE REPAIR FOR CFD:(a)BEFORE, (b)AFTER.

All the unstructured numerical grids were generated by the commercial software Gridgen. Numerical results obtained by our large-scale LES were visualized using the commercial software FieldView (Intelligent Light). The maximum data files for numerical grids and instantaneous physical values amounted to 2.7 and 1.7 Gbytes, respectively, for the visualization of a formula car. To treat such large data size comfortably, a client-server environment in FieldView was utilized.

RESULTS

ASMO model (Comparison with RANS method)

We conducted the aerodynamic assessment on the ASMO (Aerodynamisches Studien Modell) [1] vehicle body to investigate the validity of our numerical method [2]. A 1/5 wind tunnel test model is adopted and its characteristic sizes are: $T = 0.81$m(length), $B = 0.29$m(width), $H = 0.27$m (height), the wheel base is 0.54m, and the ground clearance is 0.03m. The computational domain is set to 15L/7.5B/4H. The tetrahedral elements are used and the total of 24,327,704 elements is required to cover the whole domain (4,112,586 grid nodes). The inlet mean velocity is set to $U_0 = 50.0\mathrm{m/s}$ with 0% turbulent intensity. The time increment is set to $2.0x10^{-6}$sec. For comparison with LES, we also conducted RANS using the standard $k-\varepsilon$ model on the same grid resolution as LES. In RANS, the convective term is discretized by the third order TVD scheme.

In LES, turbulence statistics are obtained for 90,000 time steps after the flow field reaches a fully developed state. In RANS, exactly the same numerical grid is used and about 10,000 steps are necessary for the simulation to be converged from the initial condition. The required CPU time for each time step is about twice or three times longer for RANS due to the implicit scheme adopted. Roughly speaking, LES is one order of magnitude more expensive than RANS in the context of CPU time to obtain the reliable turbulence statistics. Time-averaged pressure distributions along the cross-section of the vehicle normalized by the inlet pressure and velocity, $C_p = (p - p_\infty)/(\rho U_\infty^2/2)$, are plotted in Fig.1. For references, wind tunnel data using 1/5 model provided by Volvo Car are also shown. Generally both LES and RANS show good agreement with experimental results along the top surface. Some discrepancies appear along the bottom surface, and the $k-\varepsilon$ model overestimates the absolute pressure at the front region ($0.1 < x/L < 0.25$), while LES shows good agreement with wind tunnel data. The poor estimation of the pressure drop at x/L=0.8 (on the edge of the diffuser part) by the $k-\varepsilon$ model is also clearly improved by LES. The excellent agreement between LES and experimental data is remarkable at the back-end of the vehicle, while the $k-\varepsilon$ model slightly overestimates the absolute base pressure. It was discussed that reproduction of the base pressure is difficult by the standard $k-\varepsilon$ model[3].

Motorcycle, SUZUKI GSV-R (Wrapping technique for the pre-processing)

When we treat CFD as a computer aided engineering (CAE) tool, not only the accuracy of the numerical method adopted, but also the cost or the user-friendliness of pre- and post-process is important. In fact, such a target with a complicated geometry as automobiles, the cost or man-hour for the pre-processing, including the adjustment and repair of CAD data, simplification of surface geometry by the wrapping technology, as well as the grid generation, is comparable with that for the simulation itself even in LES.

The wrapping technique is especially useful for the simulation of a motorcycle, which has a tediously complicated shape. As a typical example, the validity of the CAD repair technique is demonstrated in Fig.2. We used a commercial software 3Matic(by Materialize N.V.,Leuven,Belgium / Materialize Japan Co.,Ltd.) to repair the raw CAD data and to optimize the numerical mesh for CFD. Simplification of surface geometry makes it possible to conduct LES of flow around the motorcycle using the numerical elements of no more than 30 million. The simulation was conducted using about 28 million numerical elements (tetrahedron) on 32 nodes (256 CPUs). Snapshots of the unsteady vorticity distribution reproduced by our LES and resultant distribution of pressure fluctuation on the surface of the motorcycle and a rider are shown in Figs.3, and 4, respectively.

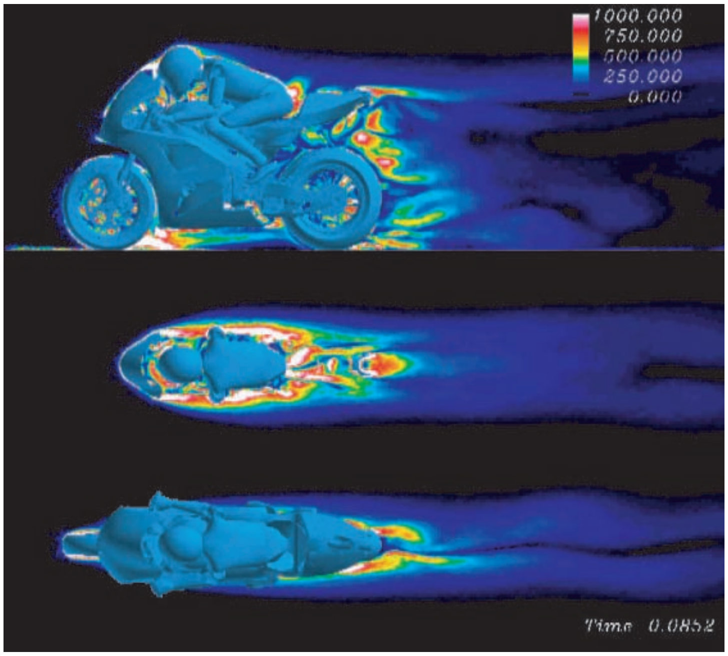

Figure 3. SNAPSHOTS OF VORTICITY DISTRIBUTION AROUND THE MOTORCYCLE.

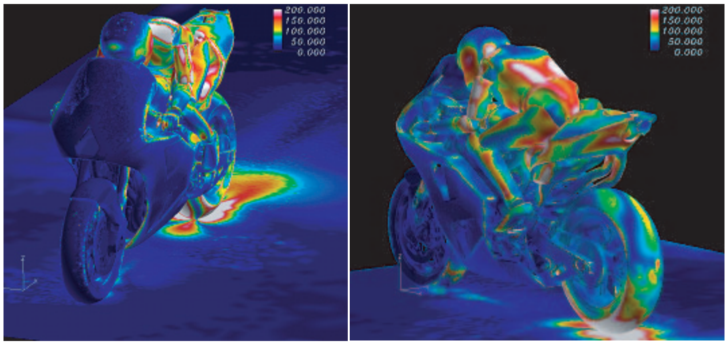

Figure 4. SURFACE PRESSURE FLUCTUATION OF THE MOTORCYCLE.

Commercial Sedan, MAZDA ATENZA (Reproduction of unsteady eddies)

The HPC-LES we have developed was successfully applied also to a commercial sedan involving the engine room and un-

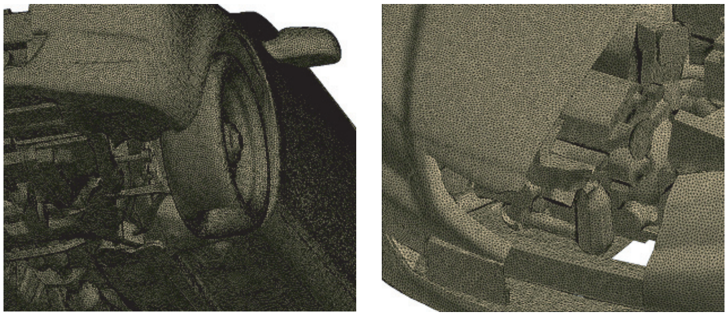

Figure 5. SURFACE MESH CONFIGURATIONS OF THE COMMERCIAL SEDAN.

derbody with complicated shapes, which is shown in Fig.5. The simulation was conducted using about 38 million numerical elements (tetrahedron) on 64 nodes (512 CPUs). Recently the importance of the aerodynamics of vehicle underbody is pointed out [3]. As shown in Fig.6, the flow under the body unsteadily interacts with the vehicle wake, which will strongly affects the total aerodynamic performance of the vehicle. It also can be identified that part of the underbody flow comes from the engine room, thereby inclusion of the engine room in the simulation is indispensable, even though it costs more computer resources.

Fig.7 indicates the 3-D velocity distribution around the sedan. Characteristic eddy structures can be found such as at the rearward of the front wheel, the wake of the side mirror, the backward of the A-pillar, and above the rear trunk, all of which mingle together at the backward of the vehicle forming the wake. Reproducing such eddies by LES technique is quite important to estimate the unsteady aerodynamic characteristics of the vehicle, and hence, ride comfort and driveability of the vehicle. It goes without saying that conventional RANS cannot reproduce such organized unsteady eddy structures.

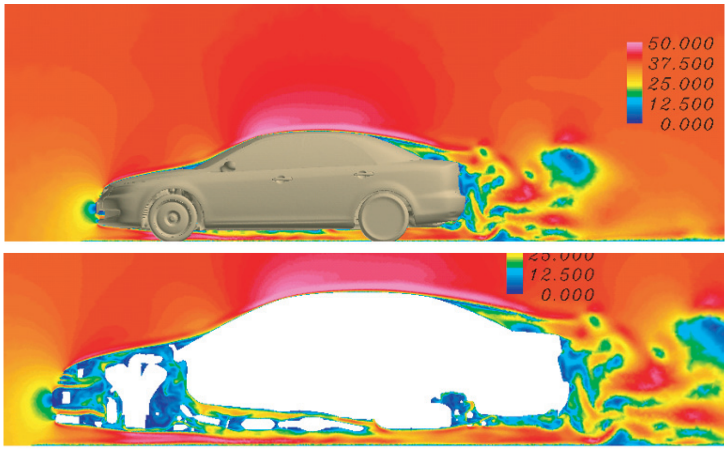

Figure 6. SNAPSHOTS OF VELOCITY DISTRIBUTION AROUND THE COMMERCIAL SEDAN.

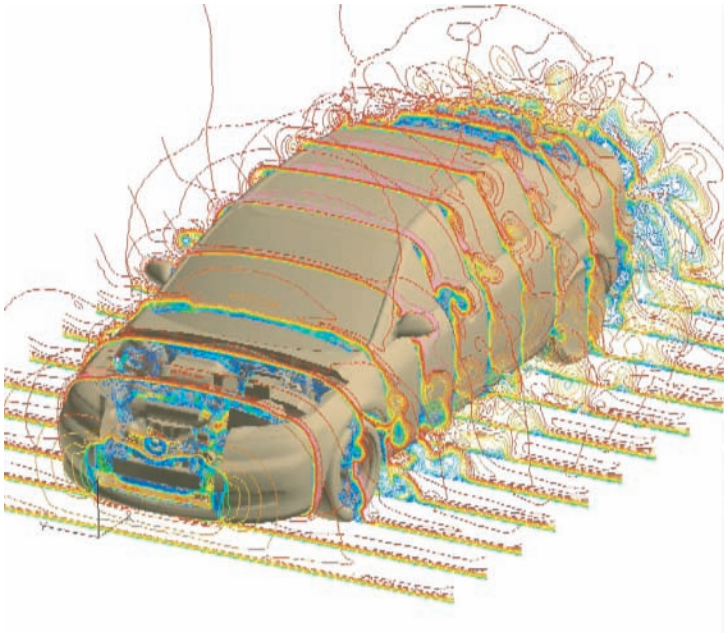

Figure 7. SNAPSHOT OF 3-D VELOCITY DISTRIBUTION AROUND THE COMMERCIAL SEDAN.

Formula Car, LOLA B03/51 (The ultimate target)

The last application introduced is a formula car [4] [5], which is one of the most severe subjects in automotive industry from the viewpoint of both the demand of very high accuracy and short time for development. The analysis model is LOLA B03/51 used in the 2006 Japanese championship Formula Nippon (former Japanese Formula 3000 Championship) season. The CAD data of the 1/2 wind-tunnel model provided by the LOLA Cars International Ltd. was preprocessed for the numerical grid generation by modifying manually the distorted or discontinuous surfaces (some hundreds places!) in the original data. The overall body size of the car is $T = 2.2$m(length), $B = 0.89$m(width), $H = 0.48$m(height), 1.50m(wheel base), 0.75m(front track), and 0.69m(rear track). The car is mounted on the floor of the rectangular computational domain of length/width/height = 34.0m/2.70m/2.47m.

On the process of grid generation, the computational domain was divided into 10 regions to treat huge amount of grids more easily especially in the context of the produced grid data file and the pre-process of the CFD software. In formula cars, it is well known that small shape distortion, or tiny aerodynamic parts in

Table 1. COMPARISON OF AERODYNAMIC FORCES OF THE FORMULA CAR WITH WIND TUNNEL TEST

	Present LES		Wind tunnel
C_d	1.00 (incl. wheels)	0.56 (excl. wheels)	0.91 (incl. wheels)
C_l	-1.68 (incl. wheels)	-1.95 (excl. wheels)	-1.93 (excl. wheels)

some cases drastically affects the total aerodynamic forces acting on the car, and the grid resolution on the surface of the car is maintained to about 2.5mm to properly reproduce such small aerodynamic devices. Accordingly total of 117,060,909 elements (20,957,323 nodes) were required to fill in the entire computational domain.

The drag and lift coefficients (C_d and C_l respectively) obtained by our LES and their comparison with wind tunnel data provided by Lola are summarized in Table 1. For the experimental reason, the provided lift coefficient does not include the effect of wheels. Agreement of the lift coefficient between the LES and experiment is excellent and our LES estimates the value only about 1% larger than the wind tunnel data, while the disagreement of the drag coefficient is relatively larger but still it is less than 10%.

Fig.8 indicates the typical examples of the interaction between the flow and the car body. Flows behind the front wheels are visualized in Fig.9. As indicated in Table 1, about 40% of total drag of the formula car is caused by the exposed wheels. Especially in the front wheels, produced wake turbulence behind the wheels strongly interacts with car body and affects its aerodynamic characteristics. Generally, wake turbulence behind the wheels are quite complicated and three dimensional, which consists of two wake features of rotating cylinder and rectangular cylinder.

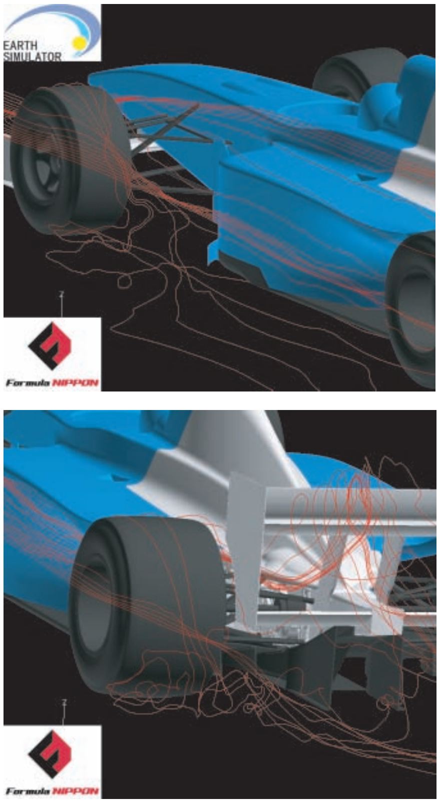

Figure 8. SNAPSHOTS OF STREAMLINES AROUND THE FORMULA CAR.

CONCLUDING REMARKS

World's largest class engineering LES of flow around vehicles was conducted on the Earth Simulator, and promising feature of HPC-LES for the assessment of vehicle aerodynamics was discussed. Excellent agreement of the pressure distribution on the surface of ASMO model between LES and the wind tunnel data could be achieved. The drag and lift coefficients estimated by our LES in the formula car was also satisfactory, which is within several percent of the wind tunnel data. In addition to its accuracy compared with RANS simulation, the validity of LES is it can capture the transient flow feature. Accordingly our next target will be to estimate the unsteady aerodynamic force acting on vehicles in the conditions of such as sudden crosswind and steering action, or overtaking. These upcoming results will provide additional demonstration that LES will be an alternative to the conventional wind tunnel testing.

ACKNOWLEDGMENT

This work is being conducted under the collaboration projects of the Earth Simulator Center, and has been supported by the projects of 'Frontier Simulation Software for Industrial Science (FSIS)' and 'Revolutionary Simulation Software (RSS21)' sponsored by Ministry of Education, Culture, Sports, Science and Technology (MEXT), Japan. This is a cooperative research with Suzuki Motor Corp., Mazda Motor Corp., JRP (Japan Race Promotion Co, Ltd.) and Le Mans Co., Ltd. The ASMO shape data provided by Volvo Car are also greatly acknowledged.

REFERENCES

[1] Aronson, D., Brahim, S. B., and Perzon, S., 2000. "On the underbody flow of a simplified estate". *SAE Paper*, pp. No. 2000–01–0485.

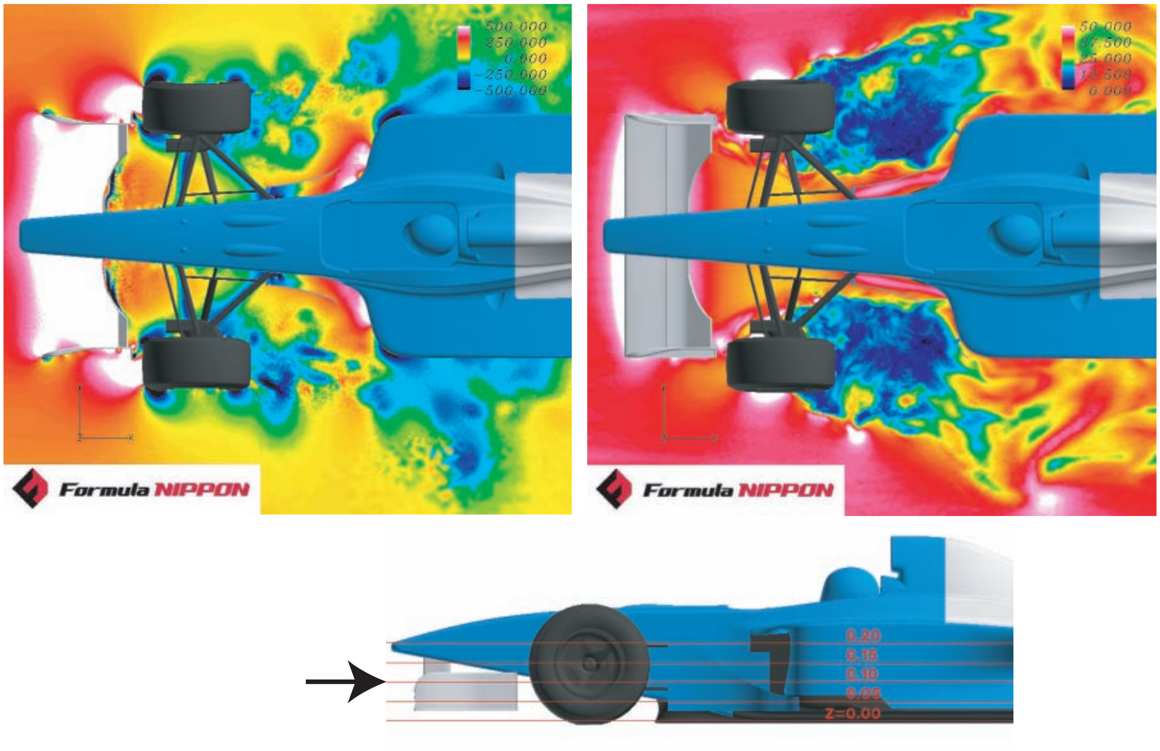

Figure 9. SNAPSHOTS OF PRESSURE(LEFT) AND VELOCITY(RIGHT) DISTRIBUTIONS AROUND THE FRONT WHEELS OF THE FORMULA CAR.

[2] Tsubokura, M., Kitoh, K., Oshima, N., Zhang, H., Onishi, K., Tominaga, T., Kobayashi, T., and Sebben, S., 2006. “High performance LES on earth simulator: A challenge for vehicle aerodynamics”. *31st FISITA World Automotive Congress*, p. F2006M111T.

[3] Kitoh, K., Kuthada, T., Oshima, N., Walker, T., and Yamamoto, M., 2006. “Aerodynamic design of the vehicle underbody: Reviews and several new computations”. *31st FISITA World Automotive Congress*, p. F2006M046.

[4] Tsubokura, M., Kobayashi, T., Oshima, N., Kitoh, K., Zhang, H., and Onishi, K., 2006. “High performance computation of unsteady flow around a formula car using large-scale LES on Earth Simulator”. *CD-ROM Proceedings of the 12th International Symposium on Flow Visualization*, p. no. 231.

[5] Tsubokura, M., Kitoh, K., Oshima, N., Nakashima, T., Zhang, H., Onishi, K., and Kobayashi, T., 2007. “Large eddy simulation of unsteady flow around a formula car on earth simulator”. *SAE 2007 world congress*, pp. 2007–01–0106.