Journal of Visualization, Vol. 11, No. 1 (2008)23-32

Review Paper

High-Performance Computing and Visualization of Unsteady Turbulent Flows

Kobayashi, T.*[1], Tsubokura, M. *[2] and Oshima, N.*[2]

*1 Japan Automobile Research Institute and University of Tokyo, Shiba-Daimon 1-1-30, Minato-ku, Tokyo 105-0012, Japan. E-mail: kobaya@jari.or.jp
*2 Graduate School of Engineering, Hokkaido University, Sapporo-shi, Hokkaido 060-8628, Japan.

Received 26 February 2007
Revised 21 December 2007

Abstract : The history of high-performance computing in turbulent flows is reviewed and their recent topics in industrial use are addressed. Special attention is paid to the validity of the method in flow visualization, and three-dimensional unsteady simulation is focused. Seemingly fundamental CFD technique for 3-D turbulence simulation has been well developed recently, but its practical use as an industrial tool has not yet become popular. An effort to close a wide gap between fundamental and practical use of scientific computer simulation is introduced through the national project promoting computational science and its development in industries of the next generation.

Keywords : HPC, LES, DNS, turbulence, CAE.

1. Introduction

As considering various fields of science and technology, computational science is promising in 21st century, which will be a basis of the technology evolution, the social safety and environmental conservation. One of the typical cases is the academic and industrial fields concerning the fluid dynamics, in which high-performance computing (HPC) has been always conducted at the latest super computers of each period. Among the applications of the computational fluid dynamics (CFD), the most notorious is turbulent flow which consists of various eddies over a wide length scale. To properly capture the characteristic feature of each turbulent flow, three dimensional (3-D) unsteady flow simulation (e.g., Shalaby et al., 2007; Fukunishi et al., 2006) must be conducted, which always uses up all CPU power of a state-of-the-art super computer. In fact until recently only the limited people could conduct such HPC in CFD. But with a rapid growth of CPU power especially for a personal computer, more reliable and realistic CFD and visualization of turbulent flow can be applied to engineering turbulence at a reasonable cost. Considering the fact that 3-D visualization of turbulent flow is still not easy in experiments because of the requirement of expensive setups with advanced technology, HPC for unsteady turbulent flow is indeed valuable for both academic and industrial use.

There are three popular methods existing in turbulence simulation: Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and Reynolds-Averaged Navier-Stokes Simulation (RANS). RANS only solves mean flow field directly with less computer resources, and thus it is the

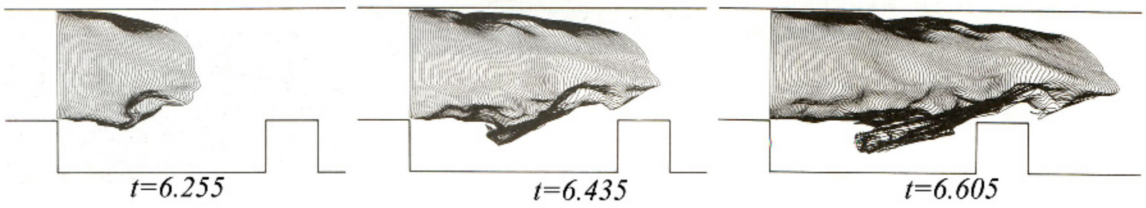

Fig. 1. Flow around turbulence promoters.

most conventional method in engineering problems at the moment. The disadvantage of RANS is it cannot treat unsteady or transient feature of turbulence, thereby results of RANS are not mentioned here. DNS and LES reproduce turbulence unsteadily in 3-D fields. DNS solves all turbulence motions directly down to the dissipative eddies, therefore its computational cost is usually quite expensive and its scope is limited to rather academic use at the moment; while on the other hand, LES treats only the larger dominant eddies and smaller eddies are modelled with less computational cost, LES is more suitable to engineering use. LES in HPC for engineering applications is mainly addressed in this paper. The review of LES development for these past thirty years is shown in sec. 2, and its recent applications for industrial use are introduced in sec. 3. Finally some concluding remarks are given in sec. 4.

2. Development of the LES Technique

2.1 The Dawn of the LES Era and the Period of Early Development

The history of LES goes back in 1970 when Deardorff first simulated a simple turbulent channel flow using only 6720 grid points. He adopted LES from the meteorological point of view and successfully found the organized eddy motions of wall-turbulence. But due to the coarse grid resolution at very high Reynolds-number assumption, agreement of turbulence statistics between the LES and experimental data was only moderate.

Compared with this simple wall-turbulence, the difficulty of engineering LES lies in the fact that shear-layer separation induced by complicated boundary geometry or inlet boundary condition at higher Reynolds-number must be resolved. Such a LES was going to be realistic in the 1980's when rapid growth of high-performance computer was going to come. But still at the period, only very limited people who could access a state-of-the-art super computer were able to perform LES. A pioneering work of engineering LES was conducted by Kano and Kobayashi in mid 1980's. They simulated a flow around turbulence promoters and successfully reproduced the large eddies generated behind a solid block. Strouhal number of the eddies estimated by the LES showed good agreement with the experimental data. The time lines (comparable to the hydrogen bubble method in experiment) obtained by their simulation are shown in Fig. 1.

The main stumbling blocks to apply LES to engineering problems at that time were concerning the boundary conditions suitable to unsteady 3-D simulation. Among them, treatment of wall boundary was, and even now is, most essential when we would like to apply LES at higher Reynolds-number flow within very limited computer resources provided. Secondary the inlet and outlet boundary conditions were also important to conduct LES stably and reliably at a restricted numerical domain. In fact development of these techniques greatly contributed to open LES technique to more researchers who have less computer powers.

The most reasonable solid wall boundary condition in CFD is ideally the no-slip condition, in which velocity is supposed to be zero on the wall. The problem of the condition is it requires many grid nodes in the vicinity of the wall to capture the very thin wall shear layer at a higher Reynolds-number condition. In 1990, Morinishi and Kobayashi developed the artificial wall boundary conditions by imposing a wall shear stress as a wall boundary condition, which is determined by a velocity at the first grid point from the wall and the Spalding's assumed mean

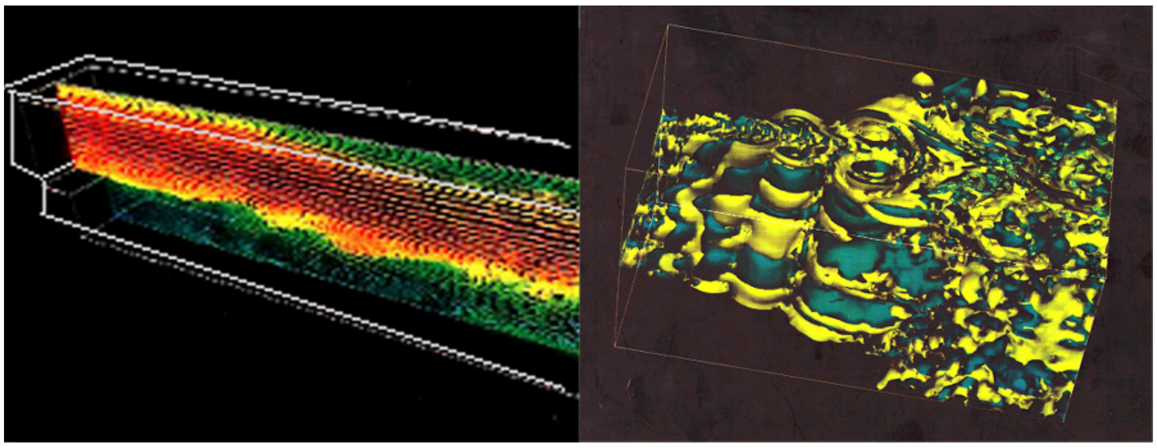

Fig. 2. Backward facing step flow (left) and planar jet (right: streamwise vorticity iso-surface).

velocity profile. They applied the method to the backward facing step flow and achieved a good estimation of reattachment length at high Reynolds-number of 40,000 using only 144,000 grid points. Later they have conducted the simulation again using 920,000 grid points to provide a database reliable to estimate and modify existing RANS models.

In 1993, Dai et al. proposed a convective outflow boundary condition including the effect of viscosity to properly capture the large eddies going out of the numerical domain stably, and conducted the LES of planar jet flow using grid points of 1,056,000. Snapshots of the backward facing step flow and the planar jet are indicated in Fig. 2. In the planar jet, the iso-surfaces of the steamwise vorticity are visualized. The roll-up eddies just below the flow inlet and the growth of spanwise perturbation at downstream are clearly observed. The rapid eddy breakdown during the transition process to turbulence is also remarkable.

2.2 The Period of Maturity

In early 1990's, LES was going to be popular for academic use (but not yet for industrial use). The main analysis objects at the period were fundamental turbulent shear flows with rather simple boundary geometry in which a Cartesian grid is available. But for more practical use with rather complicated geometry, it was indispensable to apply the generalized coordinates to LES. As a matter of fact, using the generalized coordinates in LES itself is technically not difficult, and the problem is the violation of the conservative property of momentum and/or kinetic energy in the discretized space. The problem is supposed to be critical for a reliable and stable LES/DNS. Concerning the conservative property of the finite difference scheme, Morinishi et al. (1998) proposed a fully conservative high-order accurate finite difference scheme for uniform Cartesian staggered grids and its revised version for collocate grids.

Kogaki applied the scheme of Morinishi to the generalized coordinates and conducted LES of a flow around circular cylinder and Kieda et al. (2001) utilized the method in a flow around airfoil. As shown in Fig. 3, they successfully reproduce the laminar separation bubbles with reverse flow region, which is indispensable for the estimation of the wind turbine performance.

Recently with a help of these sophisticated numerical methods as well as high performance computer, more reliable analysis of complicated 3-D turbulence structures are going to be possible in detail. In fact some of such studies pose light on the difficult problems of controlling turbulence and flow phenomena. A typical example was conducted by Tsubokura et al. (2003). It is well known that high heat and mass transfer can be achieved when a jet impinges on the wall. To investigate the relationship between the enhancement of heat transfer and eddy structures, they conducted LES/DNS of planar and round impinging jets (grid number is 4,201,600 for planar and 1,601,600 for

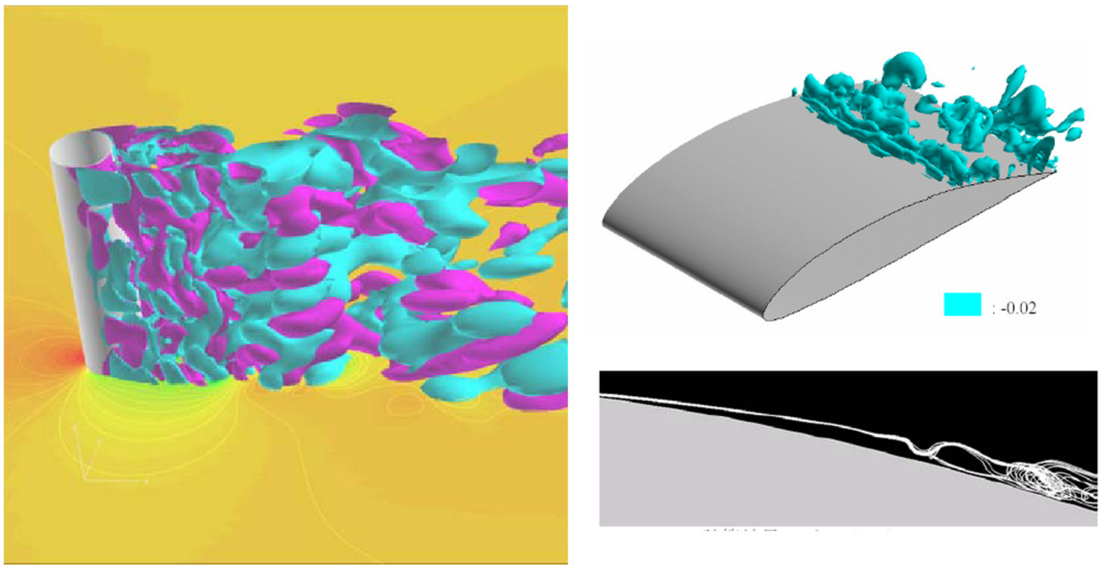

Fig. 3. Flow around a circular cylinder (left: streamwise vorticity iso-surface) and an airfoil (right: above, negative pressure iso-surface; below, particle trajectory).

round jets). They especially focused on the effect of inlet perturbation on the downstream eddy structures and tried to control the heat transfer on the viewpoint of turbulence control. Figure 4 shows the 3-D eddy structures of planar and round impinging jets near the stagnation point. The full 3-D complicated but well organized structures are reproduced by numerical simulation, which reveals that heat transfer on the wall is strongly affected by these organized motions.

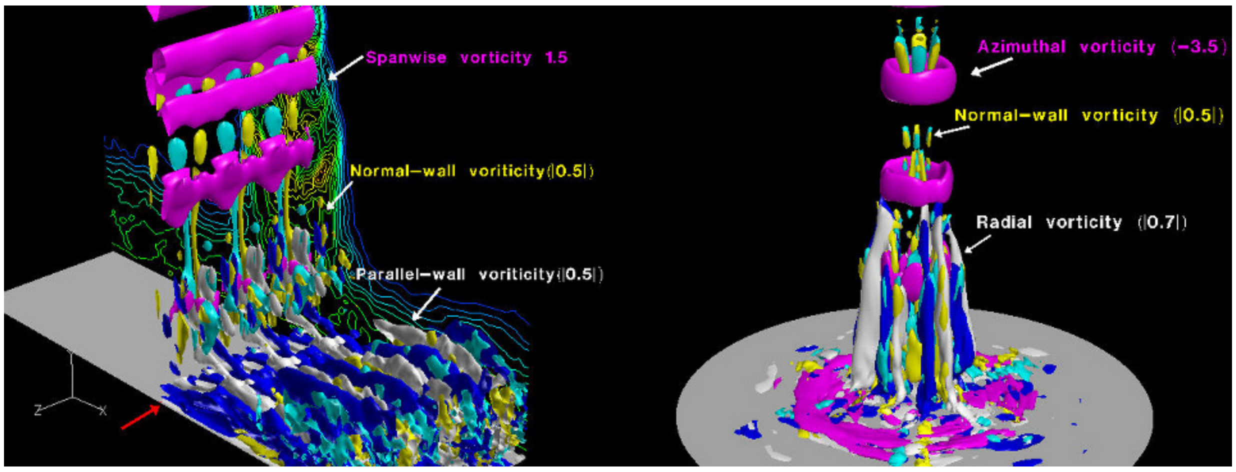

Fig. 4. 3-D eddy structures of the planar (left) and round (right) impinging jets.

2.3 Still Developing….

The basic technique of LES has already matured and the history of LES is now in the phase of more practical use in various industries where multi-scale and multi-physics phenomena such as two-phase interface, spray, chemical reaction including combustion, shock wave, and deformation or moving wall interacting with a flow. To provide reliable solutions to various problems in industries, LES technique still have to progress further in the context of such as complicated physical modeling, more sophisticated boundary conditions, as well as a numerical method itself. We will see in the next section the method to develop the LES in industrial use, and some examples of such applications.

3. Industrial Applications of HPC-LES

3.1 A Gap between Academic and Industrial Uses

There seems to have been a wide gap between the development of basic LES technique including their application to the fundamental research and their industrial use for various product designs. Nowadays, many commercial CFD tools are possible to handle LES method, and are going to be used for product design. But honestly speaking their accuracy is only moderate and they cannot stand up to be used for more critical use. In fact as noted in the previous section, LES technique has been matured but still many stumbling blocks exist for the industrial use, and the actual situation now is that each product company utilizes its own LES tools by just patching up for the moment for their own use. Therefore a cooperating project between industry and academy is required to fill the gap and to develop the LES tool which withstands industrial use.

In 2002 the project of "Frontier Simulation Software for Industrial Science" (FSIS) started at Institute of Industrial Science (IIS), the University of Tokyo, as a proposal to IT-program organized by Ministry of Education, Culture, Sport, Science and Technology (Kobayashi et al., 2002). This project investigated the five research topics of computational science, leading new generation industries. Software systems for Tera-scale computation of these scientific and engineering simulations were performed for promoting practical developments of original and fundamental researches in each field on the new generation computers and networks. "Frontier Simulation Software" was innovated by focusing intelligence of academic, industrial, and governmental circles. The project provided the standard software that lead to industrial technology by applying the frontier works of IIS and other research institutes of Japan. The project successfully ended in 2005, and inherited to the project of "Revolutionary Simulation Software" (RSS), developing the outcome of the previous FSIS project and extending to new topics.

3.2 Engineering Applications

3.2.1 Design of complex shapes

Analysis of complex flows under unsteady conditions become attractive objects for HPC. In fact, such flows have been investigated in various engineering fields with requirement of more practical predictions and designs. At the first example, a flow in the intake and compression strokes of a four-cycle gasoline engine was analyzed by Zhang et al. (2000). A model engine conditions and computation domain are indicated in Fig. 5. The grid system is generated by Multi-block method. The entire calculated domain is divided into six grid blocks and the number of total nodes was about 640,000. The information interchange between the every two blocks is carried out on "iteration-by-iteration" and "time-step by time-step" levels.

Another application was conducted by Kato (1995) for analyzing aerodynamic noise from the pantograph of the high-peed train. In this case for treating a more complicated shape, a finite element method (FEM) with around 6 million nodes was adopted for capturing the reality of their complicated shape effects. Visualization of vortical structures calculated in the LES reveals a mechanism of noise generation and contributes to the noise reduction (Fig. 6). Such simulations are also investigated for the aerodynamic designs of aircrafts and automobiles.

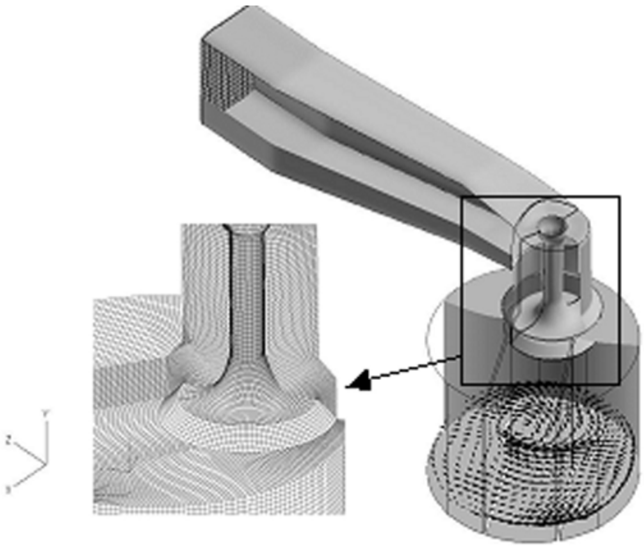

Fig. 5. Flow in the intake and compression strokes of a four-cycle gasoline engine.

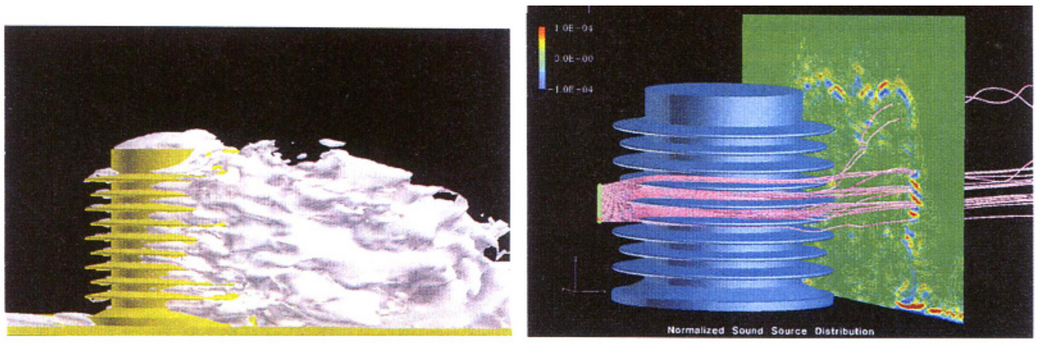

Fig. 6. Flow around a part of pantograph of high-speed train (above: instantaneous reverse flow region, below: instantaneous sound source distribution).

3.2.2 Combustion flows

As an example of power generation engineering, flow design of the industrial combustion system is introduced (Itoh et al., 2003). Spray combustion is widely used for industrial combustion systems such as diesel engines, furnaces, and gas-turbine combustors. The current environmental regulation requires us to achieve higher-efficiency, higher-performance and lower-emission combustor systems. An aim of this research is to develop a numerical prediction tool for turbulent spray combustion flows with LES, which is to be used for a systematical design of the clean and high-efficient combustion equipment.

Figure 7 shows the highly unsteady air flow and the reversed flow region at the center of the inlet nozzle caused by the existence of the swirl are observed. The reversed flow decreases from the reactive force of the droplet drag where the spray droplets exist. In the mixture fraction distribution, the fuel gas generated by spray droplet vaporization is blended with the surrounding air flow while the mixture is transported to downstream in the chamber. In contrast, we can see that another vaporized fuel gas is accumulated in the reversed flow region.

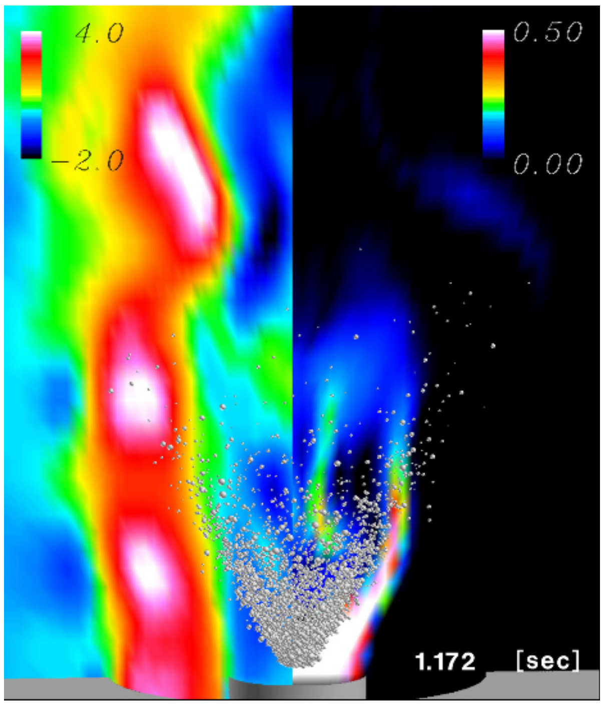

Fig. 7. Spray Combustion (left:axial air-flow velocity distribution, right: mixture fraction distribution; and individual droplet motion (spheres)).

3.3 High-End Computing for Vehicle Aerodynamics

The world's largest class unsteady turbulence simulation of flow around a formula car (LOLA B03/51) was recently conducted using LES on the Earth Simulator in Japan (Tsubokura et al., 2007). The simulation was conducted under the RSS project (see sec. 3.1). The unstructured Finite Volume software "FrontFlow/red" developed at the Univ. of Tokyo under the project of FSIS was intensively optimized for the execution on the Earth Simulator. We could finally achieve the high vectorization and parallelization of more than 95 % and 99 %, respectively, in the simulation on 100 nodes and 800 parallel processors, which made it possible to conduct the unsteady turbulence simulation around the formula car using 120,000,000 meshes!

The main objective of the study was to investigate the validity of LES, as an alternative to a conventional wind tunnel measurement, for the assessment of vehicle aerodynamics. 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 economical efficiency. However, the RANS commonly used in CFD has two fundamental 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. In particular, greater attention is paid recently 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. The problem is more critical in a race car, and an alternative method to the conventional manners is strongly desired. 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.

Figure 8 shows the instantaneous (left-top) and time averaged (left-bottom) velocity distributions on the cross section, and the instantaneous vorticity distributions (right). Unsteady characteristics of the flow are significant especially at the wake region of the car. Time depending flow structures are

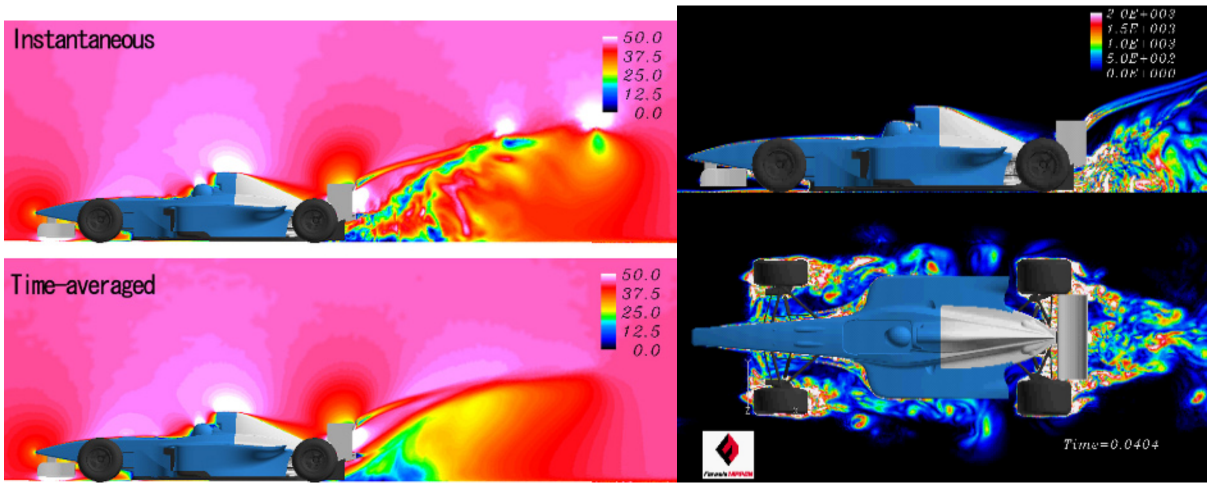

Fig. 8. The instantaneous and time-averaged velocity distributions on the cross section (left) and the instantaneous vorticity distributions (right).

Fig. 9. The Gurney flaps on the trailing edge of a rear wing (left) and the flow reproduced by HPC-LES (right).

also generated around the front tires, which strongly interact with the vehicle body. The reason why HPC with more than 100 million elements is needed for the aerodynamic estimation of a formula car comes from the fact that very tiny aerodynamic parts sometimes drastically improve the aerodynamic characteristics, and the typical example is the Gurney flaps shown in Fig. 9 (left). The Gurney flaps are mounted at the trailing edge of the front and rear wings, the height of which is some mm and consequently very fine grid resolution is required to properly capture flow around the flap. Snapshot of the flow around the flap reproduced by the simulation is shown in Fig. 9 (right). It is widely acknowledged that the flap introduces a vortex street and the vortex shedding enhances the base suction of the wing. We can clearly identify in the figure the wake produced by the flap and corresponding vortex shedding. The aerodynamic forces acting on the car obtained by our simulation show good agreement with wind tunnel data, and the lift coefficient is estimated only 1 % higher than the experimental data.

As a result, it was shown that LES in HPC could be an effective turbulence model in the very near future for the computations of the flow around the vehicle with complex configurations.

4. Concluding Remarks

History of HPC in 3-D unsteady turbulent flow and their recent application in industrial problems are reviewed. CFD is now considered to be a very strong tool for studying the turbulence in an academic use. But contrary to a vast demand for a scientific computer simulation in various engineering fields, its actual contribution in industrial problems or tools is only limited. In fact there seems to be a wide gap between the academic use in fundamental research and practical use in industry, and many stumbling blocks must be removed for CFD to be used as industrial tools. The key must be how CFD can reliably resolve the multi physics (combined problems of flow and noise, thermal, structure) and multi-scale dynamics (chemical reaction, two-phase interface, spray etc.) in the near future. Ostensibly CFD seems to be very matured but many problems must be solved for industrial use, and concerning this matter many challenging topics await CFD researchers (Kobayashi and Tsubokura, 2007).

References

Dai, Y., Kobayashi, T. and Taniguchi, N., Large eddy simulation of plane turbulent jet flow using a new outflow velocity boundary condition, Int. J. of JSME ser. B, 37 (1994), 242-253.

Deardorff, J. W., A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers, J. Fluid Mech., 41 part 2 (1970), 453-480.

Fukunishi, Y., Sakai, T., Sasaki, K., Izawa, S. and Xiong, A. K., Feedback Control of Instability Waves in a Transitional Flat-Plate Boundary Layer, Journal of Visualization, 9-3 (2006), 283-290.

Itoh, Y., Taniguchi, N., Masaki, K. and Kobayashi, T., Numerical Prediction of Vaporizing Spray Behavior by using Large-Eddy Simulation in Swirling Flows, Procs. of 5th Asian Computational Fluid Dynamics (ACFD5), (2003), 636-643.

Kano, M. and Kobayashi, T,, Study on numerical prediction of the flow around turbulence promoters –Formation of streaklines by large eddy simulation, SEISAN-KENKYU (Month. J. of Institute of Industrial Science, Univ. of Tokyo), 36-12 (1984), 24-27.

Kato, C., Numerical simulation of aerodynamic sound radiation from low Mach number turbulent wakes, ASME FED, 219 (1995), 53-58.

Kieda, K., Kogaki, T., Matsumiya, H., Taniguchi, N. and Kobayashi, T., Numerical Simulation of 3-Dimensional Flow Fields around an Airfoil for Wind Turbine, 20th ASME Wind Energy Symposium, AIAA-2001-0057, (2001).

Kobayashi, T. and Tsubokura, M., Current Status on Large-Eddy Simulation for Engineering Applications, Journal of Visualization, 10-2 (2007), 149-152.

Kobayashi, T., Taniguchi, N. and Kato, C., CFD software based on Large Eddy Simulation – Development of Simulation Software for Frontier Fields of Science and Technology, Research Revolution 2002, J. Japanese Soc. Comp. Fluid Dynamics, 10-2 (2002) (in Japanese).

Morinishi, Y., Lund, T., Vasilyev, O. V. and Moin, P., Fully conservative higher order finite difference schemes for incompressible flow, J. Comput. Phys., 143 (1998), 90-124.

Morinishi, Y. and Kobayashi, T., Large Eddy Simulation of Backward Facing Step Flow, Engineering turbulence modelling and measurements, (1990), 279-286, Elsevier Science.

Shalaby, H., Janiga, G., Laverdant, A. and Thevenin, D., Turbulent Flame Visualization Using Direct Numerical Simulation, Journal of Visualization, 10-2 (2007), 187-196.

Tsubokura, M., Kitoh, K., Oshima, N., Nakashima, T., Zhang, H., Onishi, K. and Kobayashi T., Large eddy simulation of unsteady flow around a formula car on earth simulator, SAE 2007 World Congress, (2007), paper No. 2007-01-0106.

Tsubokura, M., Kobayashi, T., Taniguchi, N. and Jones W. P., A numerical study on the eddy structures of impinging jets excited at the inlet, Int. J of Heat and Fluid Flow, 24 (2003), 500-511.

Zang, H., Kobayashi, T. and Taniguchi, N., Large Eddy Simulation of Motored Engine, FISITA World Automotive Congress, (2000), paper No. A008.

Author Profile

Toshio Kobayashi: He received his Ph.D. in Mechanical Engineering Department, the University of Tokyo in 1970. After completion his Ph.D. program, he has been faculty member of the Institute of Industrial Science (IIS) University of Tokyo, and since 1986 was a professor of Fluid engineering at University of Tokyo. In 2002, he was leader of National Project of "Frontier Simulation Software for Industrial Science". Currently he is a member of Science Council of Japan and President of Japan Automobile Research Institute (JARI). His research interests are numerical analysis of turbulence, especially Large Eddy Simulation and Particle Image Velocimetry. He serves as the President of Visualization Society of Japan (VSJ), Executive Vice President of Automobile Engineer of Japan (JSAE) and President of the Japan Socciety of Mechanical Engineers (JSME).

Makoto Tsubokura: He received his Ph.D. in Mechanical Engineering in 1997 from the University of Tokyo. He was a visiting researcher of the department of chemical engineering at the Imperial College London (1998-1999), a lecturer of Interdisciplinary Graduate School of Science and Engineering, Tokyo institute of Technology (1999-2003), an associate professor at the department of mechanical engineering and intelligent systems, the University of Electro-Communications (2003-2007). Since 2007 he is an associate professor at the graduate school of engineering, Hokkaido University. His main research interests are the development of turbulence simulation technique and its application to engineering problems. Currently he is interested in the development of vehicle aerodynamic simulator for the next generation.

Nobuyuki Oshima: He received his PhD (Eng) in Marine Engineering in 1989. He worked in in University of Tokyo as a lecturer (1989-1991), an associate professor (1991-2000) of Institute of Industrial Science and of Center for Information Technology, (2000-2005), then became a professor of Division of Mechanical and Space Eng., Hokkaido University. His current research interests are computational fluid dynamics, especially for simulaiton of turbulence and combustion.