

Chapter 12

Complex Phenomena Unified Simulation Research Team

12.1 Members

Makoto Tsubokura (Team Leader)

Keiji Onishi (Postdoctoral Researcher)

Chung-gang Li (Postdoctoral Researcher)

Leif Niclas Jansson (Postdoctoral Researcher)

Rahul Bale (Postdoctoral Researcher)

Tetsuro Tamura (Visiting Researcher)

Ryoichi Kurose (Visiting Researcher)

Gakuji Nagai (Visiting Researcher)

Kei Akasaka (Visiting Researcher)

12.2 Research Activities

The objective of our research team is to propose a unified simulation method of solving multiple partial differential equations by developing common fundamental techniques such as the effective algorithms of multi-scale phenomena or the simulation modeling for effective utilization of the massively parallel computer architecture. The target of the unified simulation is supposed to be complex and combined phenomena observed in manufacturing processes in industrial circles and our final goal is to contribute to enhance Japanese technological capabilities and industrial process innovation through the high-performance computing simulation.

Most of the complex flow phenomena observed in manufacturing processes are relating to or coupled with other physical or chemical phenomenon such as turbulence diffusion, structure deformation, heat transfer, electromagnetic field or chemical reaction. While computer simulations are rapidly spreading in industry as useful engineering tools, their limitations to such coupled phenomena have come to realize recently. This is because of the fact that each simulation method has been optimized to a specific phenomenon and once two or more solvers of different phenomena are coupled for such a complicated target, its computational performance is seriously degraded. This is especially true when we utilize a high-performance computer such as K-computer. In such a situation, in addition to the fundamental difficulty of treating different time or spatial scales, interpolation of physical quantities like pressure or velocity at the interface of two different phenomena requires additional computer costs and communications among processor cores. Different mesh topology and hence data structures among each simulation and treatment of different time or spatial scales also deteriorate single processor performance. We understand that one of the keys to solve these problems is to adopt unified structured mesh and data structure among multiple simulations for coupled phenomena. As

a candidate of unified data structure for complicated and coupled phenomena, we focused on the building-cube method (BCM) proposed by Nakahashi[1].

[1]K. Nakahashi, High-Density Mesh Flow Computations with Pre-/Post-Data Compressions, Proc. AIAA 17th CFD Conference (2005) AIAA 2005-4876

12.3 Research Results and Achievements

12.3.1 Development of a unified framework for large-scale multiphysics problems

Based on the Building Cube Method (BCM), we have developed a unified solver framework CUBE (Complex Unified Building cubeE) for solving large-scale multiphysics problems. The framework has a modular design where CUBE provides a core library containing kernel functionalities e.g. a mesh, flow fields and I/O routines. Solvers are then developed on top of the kernel by connecting necessary kernel modules together, forming a solver pipeline, describing the necessary steps to solve a particular problem.

Load balancing is an essential component in today's large scale multiphysics simulations, and with an ever increasing amount of parallelism in modern computer architecture it is essential to reduce even the slightest workload imbalance. An imbalance could severely impact an application's scalability. Traditionally, load balancing is seen as a static problem, closely related to the fundamental problem of parallel computing, namely data decomposition. For a CFD simulation based on BCM, since each cubes contains the same amount of cells the goal is to evenly distribute the cubes among the available cores. However, such a decomposition assumes that the workload for each cube is uniform. For most cubes this is true, but for cubes which contain immersed bodies, combustion, chemical reactions, etc. the workload is slightly higher, which implies a workload imbalance. Therefore, to retain good scalability a load balancing method that balances the workload not only considering the BCM mesh, but also the additional workload from the immersed body, chemical reactions, etc., was developed.

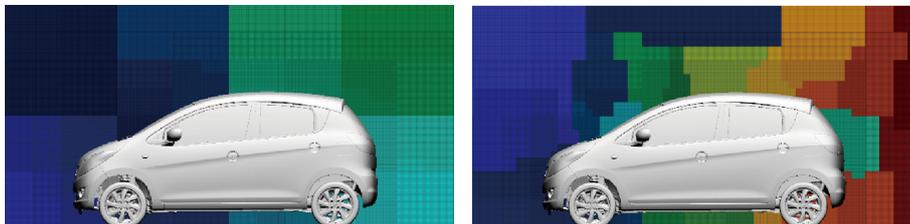


Figure 12.1: An example of load balancing with respect to the cost of evaluating the immersed geometry and the cost of computing the fluid cells, colored by MPI rank.

To evaluate the performance of the load balancer, we used CUBE to solve two different incompressible flow problems (full vehicle and a landing gear model) on the K computer. And, the total execution time for performing a fixed number of time steps for both an unbalanced (no load balancing) and a balanced case (using load balancing) on various numbers of cores are compared (Figure 12.2).

12.3.2 Development of a very large scale incompressible flow solver with a hierarchical grid system

The CUBE, name of our software framework, has been developed by conjunction with incompressible flow code which developed to realize the analysis in a real development process of industrial field on the massively parallel environment, including pre- and post-processing.

Industrial collaboration, MAZDA: We have utilized it for the complexed geometrical analysis using dirty CAD data received from automotive company. In last year, we have conducted basic aerodynamics validation using the vehicle geometry of MAZDA Motor Corporation. The conclusion was we need to improve an accuracy of drag force prediction. In this year, we have improved it conducting method survey of interpolation technique onto surface/volume, and the fundamental

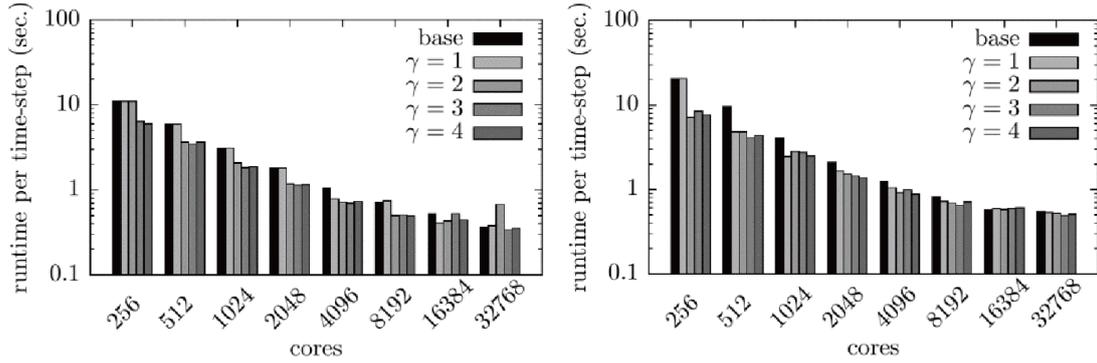


Figure 12.2: Comparison of runtime per time-step for balanced and unbalanced cases for the landing gear model (left) and the vehicle model (right)

investigation of approximate domain method on immersed boundary (IB). The difficulty was came from the uncertainty of front/back face of complicated geometry because the interpolation caused huge error if the search of face orientation has been missed. It was highly depends on the complexity of geometry and grid resolution. The method which is drawn by the analogy of Poisson solver providing a front/back face information based on flow solution has been developed. Then we could successfully get a reasonable absolute drag value (about 2% error), and drag delta between 2 different aerodynamic configurations. At the same time, we could successfully reproduce the characteristic total pressure flow field comparing to wind-tunnel measurement data.

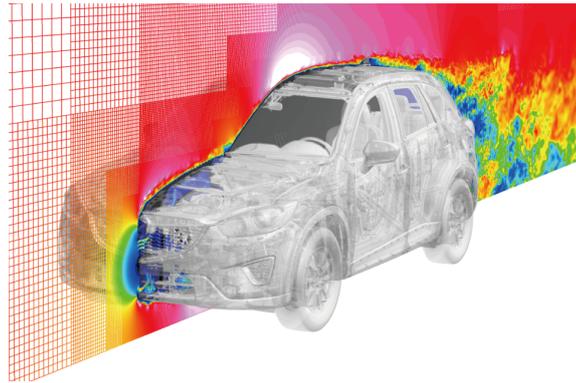


Figure 12.3: Overview of computational grid and flow field (MAZDA Motor Company).

Table 12.1: predicted drag on 2 configurations normalized by experimental base case.

C_d	Exp.	Sim.
Base	1.000	1.021
Aero	1.011	1.043
ΔC_d	1.15%	1.46%

Industrial collaboration, SUZUKI: The usability on the practical use of industrial application has been evaluated by discussing with professional engineers of automobile company. we have corroborated with SUZUKI MOTOR CORPORATION to do this work. We have provided CUBE to them with simple documents and training, and they have evaluated by their own way, and gave us their feedback report. The geometry preparation time has been accelerated from 35 hours using commercial software to 2 hours using CUBE framework, based on dirty CAD data. The estimated drag coefficient between 2 configuration had better agreement with measurement than commercial code. It shows CUBE is already ready for usage in the actual design process in term of the usability, human-cost, accuracy, and turn-around time. But, they had a strong request to improve calculation time because their process has a limitation to finish each job within 1 night whereas our method

requires 2 or 3 days. To say more, their method is based on Reynold averaged approach based on strong turbulence model which is known to have relatively large error, our method is based on the pure transient approach which generally requires 10 times larger calculation resource comparing to RANS. After the discussion, we have agreed to improve it in near future because it is very important to know the true needs on the field of industrial engineering. As the first step of that, we have developed a new function to enable us to use local refinement grids in the grid generation software. It can reduce the calculation load about from 1/5 to 1/10 for the vehicle case, to accelerate the solution. And the implementation and testing of the local refinement functionality for IB has been started in this year. We thinks it also can lead to the enhancement of effective multi-grid method (MG), or adaptive mesh refinement method (AMR) in the future development scope.

Both of MAZDA and SUZUKI could decide to promote the results inside their organization, and continue the current research activity using our software in FY2016 by submitting the application and accepted the industrial use project on K-computer.



Figure 12.4: Overview of characteristic flow field (SUZUKI MOTOR CORPORATION).

Table 12.2: predicted drag on 2 configurations normalized by experimental base case.

Condition	CUBE	Commercial Code A
Finest grid size	2.0 [mm]	2.0 [mm] (layer 0.04[mm])
Num. of cells	400 million	53 million
Fluid Configuration	LES (Standard Smagorinsky)	DES (SST $k-\omega$)
Num. of timesteps	145,000	4,000
CAD prepare	8 hours	8 hours
Pre processing time	2 hours	34.5 hours
Parallel num.	4,096 cores (K-computer)	512 cores (Intel Xeon)
Flow computation time	258 hours	8 hours
Post processing time	1.5 hours	1.2 hours
Error of Cd prediction	Applx. 10%	Applx. 11-16%
Error of ΔC_d prediction	8%	12%

Wind HPC consortium: At a research activity on Wind-HPC consortium which is organized by Tokyo Institute of Technology and several Japanese major construction companies, the detailed turbulence characteristics on wind canopy of actual urban area geometry that has housings, buildings, vegetation, and street, and so on, has been investigated. And, the academic case validation using square cylinder has been conducted. The results shows reasonable accuracy, so both of results has been published in the paper of architecture design. This research will continue on the FLAGSHIP 2020 project of priority # 4 regarding wind environment evaluation for building construction on severe climate condition through next years.

12.3.3 Development of unified compressible flow solver for unified low to moderate Mach number turbulence with hierarchical grid system

The Simulation of the low speed compressible turbulence is a key challenge for the industrial applications such as combustion, aeroacoustics and significant heat transfer phenomena. Roe scheme with a

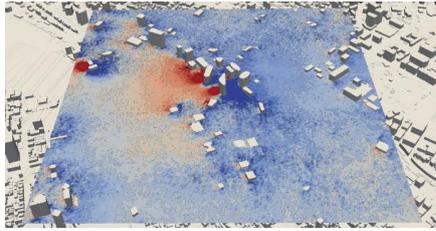


Figure 12.5: Pressure field in Shiodome area(50m Height)

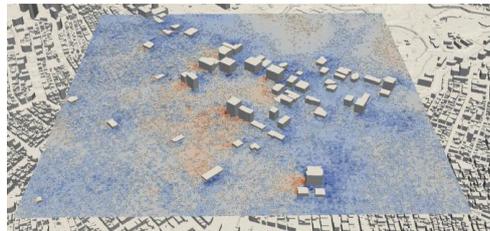


Figure 12.6: Pressure field in Marunouchi area(50m Height)

Figure 12.7: Q-criterion ($Q=0.0015$) in Shiodome areaFigure 12.8: Q-criterion ($Q=0.0015$) in Marunouchi

low-Mach number fix [1] is adopted to tackle slow flows with variable densities. An immersed boundary method (IBM) for compressible flows with a fast, easy to implement and robust interpolation method is developed to handle the complex geometries.

Basic Validation with Academic case: Based on the experiments conducted by Jia and Gogos [2], a steady-state natural convection around a heated sphere under the condition of that, the Grashof number based on the radius of the sphere is 104, is conducted to validate the unified solver. Fig. 12.9(a) shows the contour of the velocity magnitude. The entrainment comes from the bottom of the sphere which is consistent with the description by [2]. Fig. 12.10 shows the temperature contour. Above the top of the sphere, higher temperature region is formed, which cause worse natural convection near the surface so the velocity in this region is quite low. Comparisons of the averaged Nusselt number (Nu), drag coefficients caused by pressure and viscous are tabulated in Table 12.3. The results are in good agreement with the experimental data and show the accuracy and availability of our program for dealing with the complex geometry and heat transfer problems. The present results have been published in [3].

The simulation of a sphere at $Re=104$ is performed to investigate the availability of the unified solver for higher Reynolds numbers. Fig. 12.11 shows the distribution of mean pressure coefficient. The result is well consistent with the [4] and the separation angle can be also accurately captured,

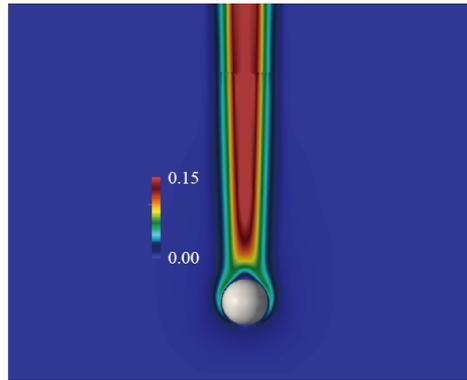


Figure 12.9: A heated sphere: velocity magnitude (m/s)

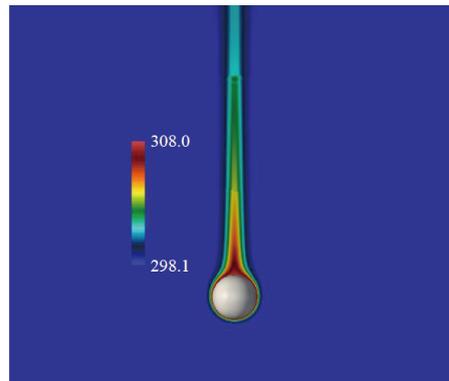


Figure 12.10: A heated sphere: temperature (K)

which is around 86 degrees.

Fig. 12.12 shows the Q criterion contoured by the magnitude of the velocity. The turbulence structures are mainly formed after the separated shear layers generated from the separation point. Besides, the transition from large to small turbulence structures can be also clearly observed in the wake region.

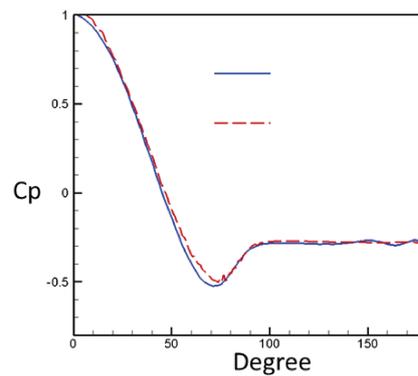
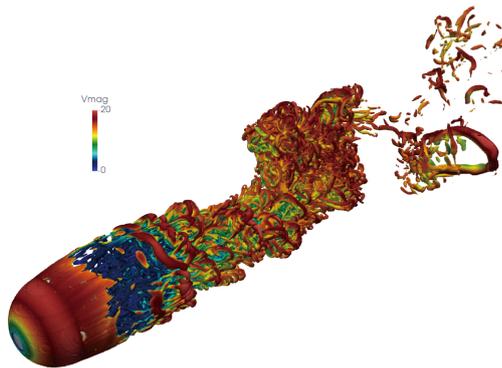


Figure 12.11: Sphere flow at $Re=10^4$: Distribution of mean pressure coefficient

Table 12.3: Comparison with existing experimental data

	Nu	$C_{D,p}$	$C_{D,u}$
Exp. [2]	8.74	0.46	0.62
Present	8.77	0.46	0.59

Figure 12.12: Sphere flow at $Re=10^4$: Q criterion

Finally, the simulation of the whole vehicle demonstrates the capability of the unified solver. Fig. 12.13 shows the contour of the velocity magnitude. The development of the boundary layer on the front window and roof can be clearly observed, which shows the capability of unified solver for handling the complex geometry. Besides, the flows penetrate the front of the car to the engine room is also obviously shown, which indicates that our immersed boundary can also treat non-watertight geometry. Fig. 12.14 shows the history of the drag and lift coefficients. After reaching the quasi steady state, the average values of them are in good agreement with the experimental data. This is an indication that the unified solver is also able to obtain accurate results for this kind of practical application. In Fig. 12.15, the Q-criterion contoured by the magnitude of velocity is shown. The development of turbulent coherent structures near the wheel, mirror and side windows is well captured. In addition, the typical turbulent structures-hair pin can be also obviously observed on the roof.

- [1] F. Rieper, *Journal of Computational Physics*, 230 (2011) 5263-5287.
- [2] H. Jia, G. Gogos, *Int. J. Heat Mass Transfer* 19 (1996) 1603-1615.
- [3] C. Li, M. Tsubokura, *Int. J. Heat Mass Transfer* 75 (2016) 52-58.
- [4] C. George, S. Kyle, *Physics of Fluids*, 16 (2004) 1449-1466.

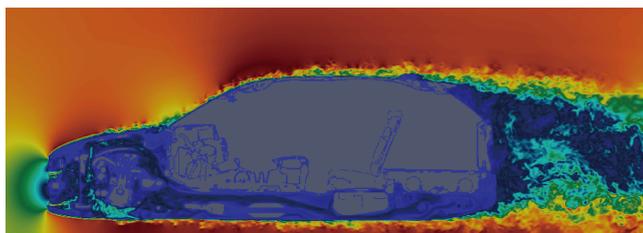


Figure 12.13: Vehicle Simulation: Snapshot of velocity magnitude

12.3.4 Development of high performance moving boundary solver for realistic motion

Aerodynamics of Vehicle in a turn: The aerodynamic performance and stability a vehicle is strongly influenced by the crosswinds during cruise and while in turning maneuvers. It is difficult to simulate such real-world flow scenarios in wind tunnel experiments. Furthermore, it is also difficult to measure unsteady aerodynamic forces in wind tunnel experiments. Thus, it is desirable for numerical methods to be able to efficiently and accurately simulate such flow conditions. To this end, here, we present simulation of a vehicle (the complex full vehicle geometry discussed in the previous section) undergoing a turning motion, including wheel rotation and turn, chassis roll and turn, in a uniform flow. This simulation the result of a collaborative work between Mazda Motor Corporation and RIKEN AICS. The detailed vehicle geometry and it motion data were provided by Mazda, and the simulation was carried out at on the K-computer using CUBE. The Lagrangian-Eulerian approach

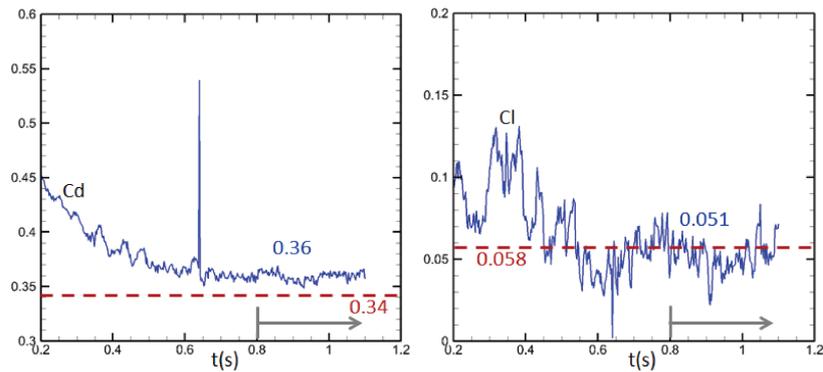


Figure 12.14: Vehicle Simulation: Time series of drag (left) and lift (right) coefficients

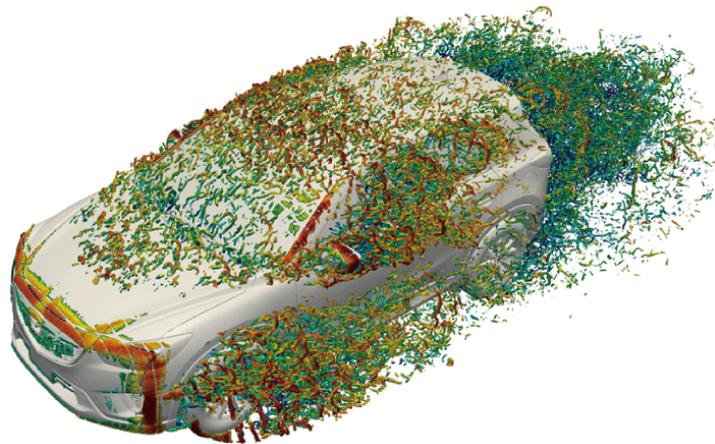


Figure 12.15: Vehicle Simulation: Snapshot of flow structures extracted by the Q-criterion

developed during the previous year was used for this simulation. As mentioned above all the vehicle motion, except linear translation, is imposed on the vehicle. If the linear translational motion was imposed on the vehicle, a fine mesh would be needed in the vehicle's path, which makes the mesh size excessively large. An alternate approach, where instead of imposing the linear translation on the vehicle it is imposed on the entire mesh, was used. In this approach the vehicle's center of gravity remains fixed relative to the mesh. So, the fine mesh is needed only in a small region around the vehicle instead of the region of the vehicle's path. This reduced the mesh size by a factor of 3-5. The results of the simulations are shown in Fig. 12.16.

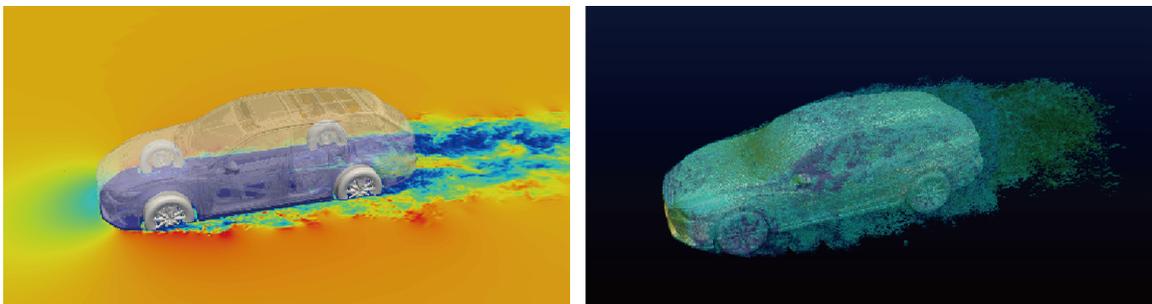


Figure 12.16: Flow field around a vehicle in a turn. (Left) Velocity magnitude on a horizontal plane. (Right) Iso surface of swirl.

Aerodynamic performance of a Ski jumper: Ski jump is a popular winter sport and is a part of winter Olympics. It is one of the sports in which Japan is competitive and has some of the best ski jumpers in the world. Ski Jump is a sport where aerodynamic interaction of the jumper and

air plays a key role in outcome of the sport. Minute changes in an athlete’s posture can go a long way, literally. The distance covered by an athlete is strongly correlated to the drag and lift forces on the athlete while in air. And, these forces are greatly influenced by the athlete’s posture. In collaboration with Prof. Keizo Yamamoto of Hokusyo University we investigated the aerodynamic performance of two of Japan’s top ski jump athletes, Haruka Iwasa and Sara Takanashi. Through the unsteady aerodynamic simulation of the two ski jumpers we analyzed the evolution of forces during a short period before and after the jump from the ski ramp. During this period the jumper changes from a sitting posture to a standing posture. Our analysis revealed that the posture and motion of Haruka Iwasa lead to lower drag force and higher lift force compared to the forces on Sara Takanashi. This is consistent with the real-world performance record of the two ski-jumpers.

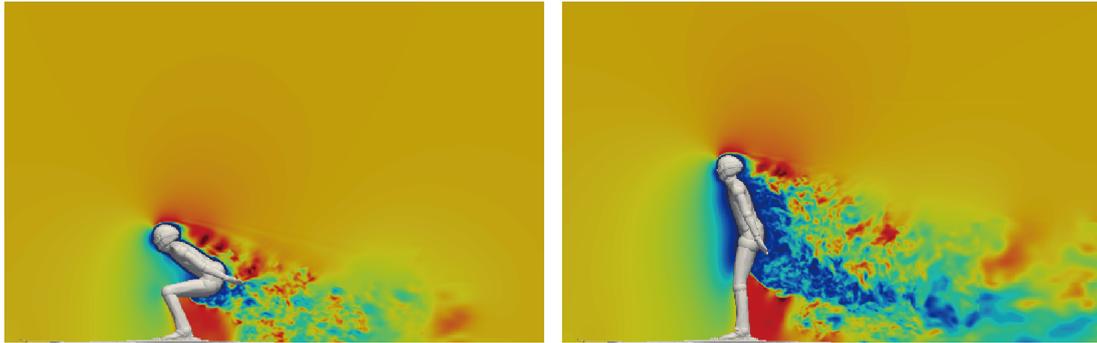


Figure 12.17: Evolution of flow around Ski jumper Haruka Iwasa during a jump. (Left) Starting posture of the jump. (Right) Final posture of the jump.

12.4 Schedule and Future Plan

(1) Five-year objectives and goals toward 2017

- Construction and development of the simulation technology for bringing out the performance of K-computer
- Proposal of the technological trend of HPC simulation toward EXA-scale

(2) Long-term objectives

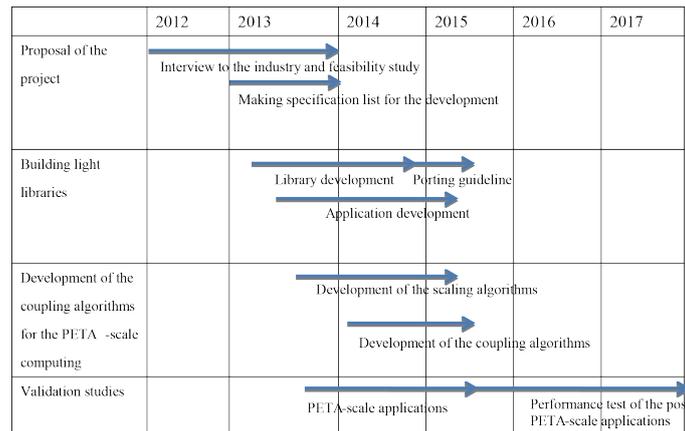
- Establishment of the research and development center for industrial simulation technology
- Contribution to computer science by expanding the developed simulation technology to different fields

(3) Time schedule

12.5 Publications

Journal Articles

- [1] Yuki Horikoshi et al. “Die design for deep drawing with high-pressured water jet utilizing computer fluid dynamics based on Reynolds’equation”. In: *Journal of Materials Processing Technology* (2015).
- [2] Chung-Gang Li, Makoto Tsubokura, and Rahul Bale. “Framework for simulating natural convection in practical applications”. In: *International Communication in Heat and Mass Transfer* (2016).
- [3] Chung-Gang Li et al. “Compressible Direct Numerical Simulation with a Hybrid Boundary Condition of Transitional Phenomena in Natural Convection”. In: *Journal of Heat and Mass Transfer* (2015).



- [4] Jing Li, Makoto Tsubokura, and Masaya Tsunoda. “Numerical Investigation of the flow around a golf ball at around the critical Reynolds number and its comparison with a smooth sphere, Flow”. In: *Turbulence and Combustion* (2015).

Conference Papers

- [5] Hirotaka Ishioka et al. “Coupled 6DoF motion and Aerodynamics Simulation of Road Vehicles in Crosswind Gusts”. In: *33rd AIAA Applied Aerodynamics Conference, AIAA 2015-3308*, pp.1-10, DOI: 10.2514/6.2015-3308 (22-26, June, 2015, Hilton Anatole, Dallas, TX). 2015.
- [6] Jing Li et al. “Large-eddy simulation for golf ball aerodynamics: The effect of surface roughness on the drag crisis and the Magnus effect”. In: *Proceeding of the ninth international symposium on turbulence and shear flow phenomena (TSFP-9)*, 7D-4, pp.1-5(June 30-July 3, 2015, The University of Melbourne, Australia). 2015.
- [7] Takuji Nakashima et al. “Unsteady Aerodynamics Simulations of a Sedan-Type Road Vehicle during Sinusoidal Steering Input”. In: *33rd AIAA Applied Aerodynamics Conference, AIAA 2015-3307*, pp.1-12, DOI: 10.2514/6.2015-3307 (22-26, June, 2015, Hilton Anatole, Dallas, TX). 2015.
- [8] Yoshihiro Okada et al. “Analysis of Wind Turbulence in Canopy Layer at Large Urban Area Using HPC Database”. In: *Aerodynamics Evaluation of Road Vehicle in Dynamic Maneuvering*, 2016 SAE World Congress (12-14, April, 2016, Cobo Center, Detroit, Michigan, USA). 2016.
- [9] Tetsuro Tamura et al. “Analysis of Wind Turbulence in Canopy Layer at Large Urban Area Using HPC Database”. In: *9th International Conference on Urban Climate (20-24, July, 2015, Toulouse France)*. 2015.

Invited Talks

- [10] 坪倉誠. リアルタイム・リアルワールド自動車統合設計システムにおける粒子法への期待. Promotech Techno Forum 2015 in NAGOYA. 2015.
- [11] 坪倉誠. リアルワールドシミュレーションが拓く新たな自動車空力. 日本自動車技術会2015年春季大会フォーラム. 2015.
- [12] 坪倉誠. 京からポスト京へ 大規模CFDが拓く新たな応用空力シミュレーション. 第29回数値流体力学シンポジウム. 2015.
- [13] 坪倉誠. 自動車周りの非定常空気力解析. 日本機械学会No.15-33講習会. 2015.

Posters and Presentations

- [14] Rahul Bale et al. *Immersed boundary method in a multiphysics framework for large scale simulations*. 27th International Conference on Parallel CFD, (May 17-20, Montreal, Quebec, Canada). 2015.
- [15] Niclas Jansson. *Towards large-scale multiphysics simulations on the K computer*. BIT Circus 2015 , (August 26-27, Umea Sweden). 2015.
- [16] Niclas Jansson and Makoto Tsubokura. *Dynamic Load Balancing for Large-Scale Multiphysics Simulations*. 第29回数值流体力学シンポジウム, 博多 日本 2015.
- [17] Hidenori Kawai et al. *HPC-Based LES of Wind Flow Over Large Urban Area with Slight Undulation*. ECCOMAS Congress 2016, (June 5 - 10, 2016, Crete Greece). 2016.
- [18] Chung-Gang Li, Tsubokura Makoto, and WuShung Fu. *An Investigation of Transitional Phenomena from Laminar to Turbulent Natural Convection using Compressible Direct Numerical Simulation*. 3rd International Workshop on Heat Transfer Advances for Energy Conservation and Pollution Control, (October 16-19, 2015, Taipei,Taiwan). 2015.
- [19] Chung-Gang Li and Makoto Tsubokura. *A New Program for the Application of Natural Convection in Practical Products*. Twelfth International Conference on Flow Dynamics, (October 27-29, 2015, Sendai, Japan). 2015.
- [20] Chung-Gang Li and Makoto Tsubokura. *A New Simulation Framework for the Natural Convection in Practical Products*. American Physical Society 68th Annual DFD meeting, (November 22-24, 2015, Boston, USA). 2015.
- [21] Chung-Gang Li and Makoto Tsubokura. *A Unified Simulation Framework for Low Speed Compressible Turbulence in Industrial Applications*. 28th International Conference on Parallel CFD, (May 9-12, Kobe, Japan). 2016.
- [22] Chung-Gang Li and Makoto Tsubokura. *An investigation of transitional Phenomena from Laminar to Turbulent*. 15th European Turbulence Conference 2015, (August 25-28, 2015, Delft, Netherlands). 2015.
- [23] Chung-Gang Li et al. *A Unified Simulation Framework for Compressible Flows at All Speeds in Industrial Applications*. 第29回数值流体力学シンポジウム, 福岡 日本 2015.
- [24] Keiji Onishi and Makoto Tsubokura. *Parallel Computation of Sliding Mesh for the Unstructured Grid Finite Volume Method on the Rotating Wheel Vehicle Aerodynamics Simulation*. 28th International Conference on Parallel CFD, (May 9-12, Kobe, Japan). 2016.
- [25] Keiji Onishi, Makoto Tsubokura, and Takashi Kamioka. *Practical Full Vehicle Multi- Physics Aerodynamics Simulation for Engineering Design Process Using Large-scale Parallel CFD Framework*. ECCOMAS Congress 2016, (June 5 - 10, 2016, Crete Greece). 2016.
- [26] Bale Rahul and Makoto Tsubokura. *Lagrangian-Eulerian based immersed boundary method for large scale simulations*. BIT Circus 2015 , (August 26-27, Umea Sweden). 2015.
- [27] Bale Rahul et al. *Constraint Based Immersed Boundary Method for “Thin” Structures*. 28th International Conference on Parallel CFD, (May 9-12, Kobe, Japan). 2016.
- [28] Bale Rahul et al. *Lagrangian-Eulerian based immersed boundary method for large scale simulations*. 第29回数值流体力学シンポジウム, 博多 日本 2015.
- [29] Bale Rahul et al. *Scalable Immersed Boundary Method for Large scale Simulations with Moving Immersed Structures*. ECCOMAS Congress 2016, (June 5 - 10, 2016, Crete Greece). 2016.
- [30] Tetsuro Tamura et al. *High performance computation by BCM-LES on flow and pressure field around buildings*. 8th International Colloquium on Bluff Body Aerodynamics and Applications, (June 7 - 11, 2016, Boston USA). 2016.
- [31] Tetsuro Tamura et al. *LES for wind turbulence in canopy layer at large urban area*. 68th Annual Meeting of the APS Division of Fluid Dynamics, (November 22-24, 2015, Boston USA). 2015.
- [32] 大西慶治, 坪倉誠. *Dirty CAD形状を意識した自動車空力シミュレーション手法開発*. 第21回計算工学講演会, 新潟 日本 2016.
- [33] 大西慶治, 坪倉誠. *DirtyCADデータへ対応した並列IB-BCMによる自動車空力シミュレーションの実用性評価*. 第29回数值流体力学シンポジウム, 博多 日本 2015.

- [34] 大西慶治, 坪倉誠. *irtyCAD*データへ対応した並列*IB-BCM*による自動車空力シミュレーションの実用性評価. 第29回数値流体力学シンポジウム, 博多 日本 2015.
- [35] 赤坂 啓, 藤山 祐司, 大西 慶治 and 坪倉 誠. 自動車操縦安定性および空力性能の連成に関する非定常空力解析手法の構築. 第2回「京」を中核とするHPCIシステム利用研究課題 成果報告会, 東京 日本 2015.