HIGH-FIDELITY NUMERICAL SIMULATIONS FOR DESTRUCTIVE RE-ENTRY OF UPPER STAGES

Keiichiro Fujimoto, Hiroumi Tani, Hideyo Negishi, Yasuhiro Saito, Nobuyuki Iizuka, Koichi Okita

Japan Aerospace Exploration Agency, 3-1-1, Sengen, Tsukuba, Ibaraki, Japan, fujimoto.keiichiro@jaxa.jp

ABSTRACT

In order to keep our space activity sustainable, comprehensive research effort should be taken for the space debris problem. Improvement in the destructive re-entry risk analysis methodology is one of the important research areas to reduce the uncertainties on the expected casualty predictions and to realize the design-for-demise. Uncertainty quantification strategy was proposed, and the role of the high-fidelity numerical simulations was discussed. Based on the detailed investigation of the computed results, selection of the reference length to predict the stagnation heat flux, considerations on the local curvature and the shock-interaction effect were found to be the key factors to achieve high-order accurate heat flux predictions.

1 BACKGROUND AND OBJECTIVES

Space debris problem is a growing concern to be tackled internationally to keep our space activity sustainable. Numerous pieces of debris are tracked on the orbit around the Earth, those number will be rapidly increased due to the growing satellite demands such as mega-constellations. The increasing number of space debris directly results in the increasing potential risk to all spacecraft. Therefore, a comprehensive research and development (R&D) efforts should be made to understand the space debris environment, assess its risk, mitigate its number growth, and control the risk. An importance to ensure a sustainable space environment has been firstly stated in space basic act of Japan in 2008. One of the major plans on the space policy is to address the R&D activities for the space debris problems. Reorganizations of the regulation issues and R&D activities for space debris of Japan Aerospace Exploration Agency (JAXA) were initiated in 2016. Four major activities were identified and re-organized, those are 1) formulation of space debris international standards and guidelines, 2) debris situational awareness and defense, 3) low cost active debris removal, and 4) debris mitigation [1, 2].

Space debris mitigation is the most efficient strategy to reduce the number of orbital debris in the critical orbital zones for long term sustainability. There are two mitigation approaches to remove the derelict spacecraft from the protected regions, the one is transfer to the non-protected orbit and the other is the destructive atmospheric re-entry. Risk should be minimized as much as possible by the comprehensive consideration on the design and the disposal operation based on the mission analysis and the risk assessment. The destructive re-entry of satellites and rockets and the related ground risk due to the fragments reaching the ground are getting increased interest in recent years. Survived fragments derived from the launch vehicle's ascent and the spacecraft's re-entry have been found and retrieved in many places all over the world. Fatal accident due to artificial space debris has not been happened yet, however, the survived fragments have been found at the place near the human living area. It is apparently the strong reason to emphasize the importance to predict the risk quantitatively, and minimize the risk by the detailed consideration on the design and the end of mission disposal operations.

Therefore, focus of this study lies in the significant uncertainty reduction of the destructive re-entry safety analysis to realize the detailed design and operation considerations. The uncertainty quantification factors are identified, and effective uncertainty quantification strategy mainly based on the high-fidelity numerical simulations is proposed. Finally, an effectiveness of the high-fidelity numerical simulations is shown. Based on the investigation of the numerical results, the key findings to improve the accuracy of empirical model are discussed.

2 STRATEGY FOR UNCERTAINTY QUANTIFICATION

There are mainly two approaches for the re-entry safety analysis, the one is object-oriented approach [3] [4] and the spacecraft-oriented approach [5] [6]. Comparison and the description of these approaches can be found in present author's study [1]. There exist various uncertainty factors in the re-entry risk analysis. Major un-certainty factors identified in this study are shown in Fig. 1, those are 1) model accuracy, 2) attitude stability mode, 3) shape complexity, 4) shape change in time, and 5) initial conditions. The uncertainties on the aerodynamic and the heat flux models have significant influences on the resulting expected casualty. The empirical correlation models are used in the re-entry safety analysis [5-7]. Those models should be validated for the various geometries under the wide range of flow conditions. High-fidelity numerical simulations such as

Proc. 7th European Conference on Space Debris, Darmstadt, Germany, 18–21 April 2017, published by the ESA Space Debris Office
Ed. T. Flohrer & F. Schmitz, (http://spacedebris2017.sdo.esoc.esa.int, June 2017)

computational fluid dynamics (CFD) analysis and the direct simulation Monte Carlo (DSMC) analysis tools are validated by the comparison with the experiments under the specific flow conditions. To cover the wide range of geometries and flow conditions, the predicted results by the empirical correlation models are compared with those by CFD and DSMC. An attitude stability mode has significant effect on the time variation of the heat flux distributions, and thus heating amount. As already been clarified in the present author's previous study [1], the rocket upper stage has the possibility to have the aerodynamic stability at the pitch angle conditions where the heavier components such as engines are in the wind-ward side. As a result, the heating amount to the heavier components would be increased. Effects of the shape complexity and the shape change in time are also dominant factor. If the realistic complicated geometry can be considered in the re-entry safety analysis, it is generally resulting in the surface heating area increase. In addition, the mass, the moment of inertia, and the aerodynamic shape are also changed during the destructive re-entry of the fragments. These effects should be modelled appropriately to achieve the high-order accurate re-entry safety analysis. Furthermore, in order to realize the design-for-demise, the consideration of the realistic complicated shape is essential. Therefore, the spacecraft-oriented re-entry safety analysis method is under the development at JAXA. Initial conditions such as the temperature level, induced velocity of the components at the primary break-up are also one of the dominant factors. Those uncertainties are quantified by the flight data acquisition and the high-fidelity numerical simulations [1].

Overview of the uncertainty quantification strategy is also shown in Figure 2. Based on the correct understanding of the mechanism of the destructive re-entry, and the investigations on the uncertainty factors as described above, the physics-based models are validated on a step-by-step basis. As will be shown in the following sections, the high-fidelity numerical simulation plays important roles for the significant uncertainty reduction.

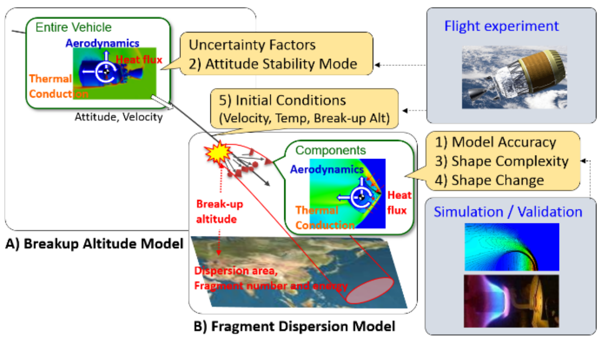

Fig. 1. Uncertainty factors for the destructive re-entry risk analysis.

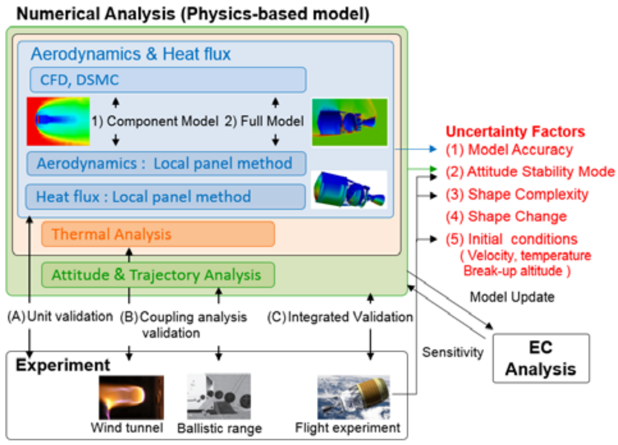

Fig.2 Uncertainty quantification strategy for destructive re-entry risk analysis.

3 HIGH-FIDELITY NUMERICAL SIMULATIONS

High-fidelity numerical simulation is the essential technology to achieve high-order accuracy in the re-entry risk analysis. Alghough the high-fidelity numerical simulations does not used in the re-entry risk analysis, it is essential to under stand destructive re-entry mechanisms and to validate the empirical correlation models used in the re-entry risk analysis. If the physical models used in the simulation are validated by comparing with the experimental data throughout the possible environmental condtions, the significant uncertainty reduction can be achieved. Based on the detailed investigation of the computed results, the mechanism of the destruction process can be understood, and the empirical correlation models used in the re-entry risk analysis can be improved by comparing with the high-fidelity numerical simulation results.

Since the accuracy of the aerodynamics and the heat flux model has significant influence on the expected casualty, the establishemnt of CFD method for the destructive re-entry analysis is essential. In the last two decades, the comprehensive and intensive research efforts have been made in order to improve CFD schemes and the computational speed especially for the aerodynamic design of the aircrafts, rockets, re-entry vehicles. Consequently, the aerodynamic and the heat flux analysis for the complicated geometry have been matured and available.

However, there are still technological gaps to establish an accurate CFD for the destructive re-entry, those are 1) model validation and accuracy improvements for the non-smooth bodies, 2) development of the capability to handle deforming complicated bodies, and 3) development of the heat flux model including important effects such as the surface chemical reactions and the shape deformations.

In this study, two state-of-the-art in-house CFD codes, named as UNITED and LS-FLOW have been employed. Those codes have superior advantages such as the capability to handle complicated geometry, and the capability of the solution adaptive mesh refinement (AMR). Furthermore, these two codes have been employed various types of the aerodynamic characteristics and the heat flux predictions, and intensive research efforts for the accuracy validaton studies have been made. Technical details on the computational fluid dynamics are described as follows respectively.

3.1 Rarefied Flow Solver (UNITED)

At the high altitude roughly over 90 km, the effecs of the rarefied gas dynamics such as non-equilibrium of thermodynamics and chemical reacions are significant. For the prediction of the surface pressure and heat flux distributions, the Boltzmann equation of kinetic theory is solved by direct simulation Monte Carlo (DSMC) method [8]. In this study, University of Tokyo Evolution DSMC (UNITED) code is employed, which has been originally developed at the university of Tokyo and has been developed by JAXA successively. UNITED has been developed and employed for the wide range of applications such as the thrustor plume interaction issues for the cargo transfer vehicle HTV [9], analysis of the aerodynamic fluctuations for the rocket upper stages.

The variable soft sphere [10] (VSS) and variable hard sphere (VHS) models [11] are available for the collision model. It also has the capability of the sub-cells [12] and chemical reactions of air [13].

Geometries of the sattelites and rocket upper stages are complicated and changed in time. The required grid resolution is strongly depending on the important flow structures, and thus flow conditions. For the computational grid, Cartesian cut cell with AMR method is employed. In addition, the capability to handle multiple bodies in 6DoF motion is available. These are key features to analyze moving multiple complicaed objects within the practical turn-around time.

3.2 Continuous Flow Solver (LS-FLOW)

At the moderate altitude roughly bellow 90 km, the flow can be treated as continuous flow. For the prediction of the surface pressure and heat flux distributions, the compressible Navier-Stokes flow solver for the arbitrary polyhedral unstructured grids, named LS-FLOW [14] is employed in this study. It is the core CFD code of JAXA, which has been applied to various flow problems including the aerodynamic analysis [15], high temerature reactive flows and the cryogenic flows [16] for the liquid rocket engines. It was initially developed by the present author [14] and has been improved continuously, various numerical schemes have been implemented [15]. LS-FLOW has been validated by comparing with the experimental data for the various types of the geometris under the wide range of flow conditions [17, 18]. Then, it has been applied to various aerodynamic design problems such as the aerodynamic analysis for the expendable and re-usable rockets and the re-entry capsules, and the capsule outer shell separation in the chaotic freestream conditions.

The system of equations are discretized and solved by the cell-centerd finite volume method based on the arbitrary polyhedral unstructured grids. Various types of the numerical schemes have been empoyed such as for the gradient evaluation of flow variables, the gradient limter functions, Euler flux, and the viscous flux. Some of the schemes have been developed in JAXA in order to establish the robust and still spatially accurate aerodynamic analysis methodology for the turbulent high Reynolds number flows with including the strong shockwaves and the massive separated flows. Various types of the turbulence models and the high-order reconstruction scheme for the unstructured grids [19] are available. Further detailed descriptions on the key features can be followd in the previous publications [18].

For the capability to handle moving multiple complicated objects, LS-FLOW has the capability to handle the arbitrary polyhedral unstructured grid, and the overset grid method with including 6DoF motion. For the computational grid, body-fitted Cartesian grid generated by LS-GRID [20] is used in this study, in which Cartesian grid is used as the volumce cells and the body-fitted layer grid is used to resolve high Reynolds number boudary layers. Since volume cells are topologically Cartesian grid, and thus the AMR method is available, which is essential to achieve practical turn-around time to consider the various types of geometries under the wide range of flow conditions.

4 DISCUSSION OF RESULTS

4.1 TURBULENT FLOW EFFECT ON HEAT FLUX

Turbulent flow effects on the heat flux level and its distribution is key factor of the uncetainty. CFD analysis of the heat flux distribution for the flat-faced round disk whose diameter is 1.665 m is carried out by LS-FLOW in this study. Two flow conditions are considered, the one is free-stream static pressure 7.99 Pa, static temperature 227 K, and flow speed of 4337 m/s, the other is static pressure 93.87 Pa, static temperature 270.65 K, and flow speed of 2810 m/s. In addition, two wall temperatures are considered 1095 K and 500 K, respectively. Computed surface heat flux and temperature distribution in the pitch symmetry plane is shown in Figure 3. Total heat flux variation on the wind-side flat surface normalized by the value at the zero pitch angle is also shown in Figure. 4. In the laminar flow computations Case1 as shown in Fig. 4, total heat flux decreases with the pitch angle. While, in the turbulent flow computations Case 2, total heat flux increases with the pitch angle. It is not shown due to the restrictions of data disclosure, the similar trend has been observed in the experimental data. It is clarified that there is significant turbulent flow effect and the pitch angle dependency on the total heat flux. In most of the conventional empirical correlation models, the surface normal direction against the free-stream, the surface curvature, and the pressure are used to describe heat flux distributions. Thus, it is clear that this total heat flux increase with the pitch angle cannot be predicted by the conventional models. It is clear that there are also Mach number dependency and the wall temperature dependency on the total heat flux as shown in Fig. 4. In addition to consider the turbulent flow effect, these dependencies also should be considered. Therefore, the comprehensive research efforts have currently been made to improve the empirical correlation model in this study.

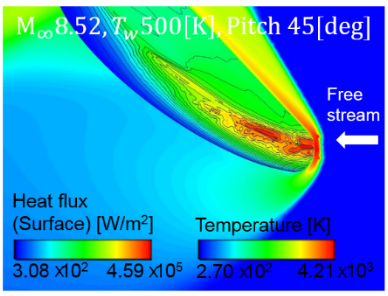

Fig. 3 Parameter depency on wind-ward mean heat flux for flat-faced body.

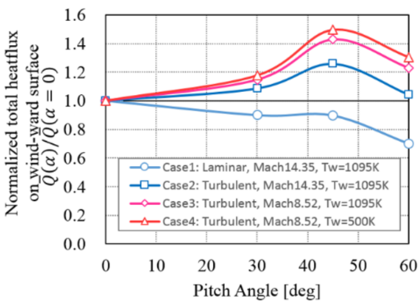

Fig. 4 Parameter depency on wind-ward mean heat flux for flat-faced body.

4.2 REAL SHAPE EFFCTS ON HEAT FLUX

As is already discussed in the previous sections, the shape of the satellites and the rocket upper stages consist of the complicated geometries and it also changes in time due to the thermal or mechanical failures. In order to establish the accurate aerodynamic and heat-flux modelling for such geometries, correct understanding of the flow structures and the appropriate formulation of the empirical correlation model are essential. Heat flux distributions over the surface for the rocket upper stage are evaluated by using LS-FLOW in this study. Heat flux distributions predicted by the empirical correlation model are compared with the CFD results. Since it is the well documented and widely used re-entry risk analysis tool, the empirical model used in SCARAB [5] is focused in this study.

Considered flow speed is 3467.8 m/s, the pitch angles are 0 and 30 degrees, and the altitude is 45 km. Maximum diameter of the vehicle is 1 meter. Computational grid is body-fitted Cartesian grid as shown in Fig. 5, which is generated by LS-GRID. Total grid cell number is 1.98 million, minimum grid size is 7.41e-6 meter in the normal directions at near surface. In the re-entry risk analysis, wide range of flow conditions should be considered. The flow structures such as separated flow, the shear layer and the shock wave are significantly changed depending on the flow conditions. Body-fitted Cartesian grid method has superior advantages to minimize the turnaround time of the analysis, since the grid resolution control is straightforward by the refinement of the cells as shown in Figure 5.

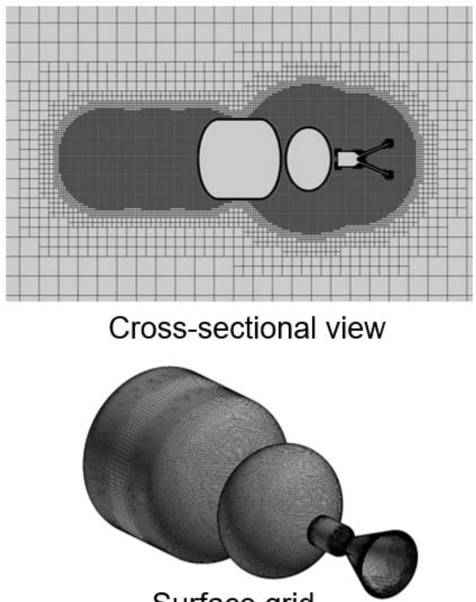

Fig. 5 Body-fitted Cartesian grid for rocket upper stage.

Computed temperature distributions in the symmetry plane and the surface heat flux are shown in Fig. 6 for pitch angle zero, and Fig. 7 for pitch angle 30 degrees. Computed surface heat flux distributions obtained by the empirical model are also shown in same figure, respectively. As shown in these figures, the flow interactions between the sub-systems are observed. At pitch zero condition, the free-stream is impinging on the LOX tank after passing through the shock wave in front of the engine. As a result heat flux level is higher at the impingement region, as comparing with the other part. Maximum magnitude of the heat flux is observed at the engine nozzle rip, which is mainly due to the small curvature radius. Since the local curvature effect is not considered to predict the heat flux distributions in the formulation of the empirical model employed for SCARAB, heat flux level at nozzle rip of empirical model is smaller than that of CFD. On the other hand, the free-stream is directly impinging on the LOX and LH2 tanks at angled conditions as shown in Figure 7. As a result, it is clarified that the heat flux level for angled condition is larger thant that for zero pitch angle. Computed heat flux distributions by the empirical model are also shown in Figure 7. Although, the free-stream is directory impinging on the tanks, and thus the flow interaction effects between the sub-systems are smaller than that for the zero pitch angle condition. For the evaluations of stagnation heat flux, the referene length derived from the total projection area of three sub-systems. This stagnation heat flux is commonly used for the heat flux evaluations for all of the sub-systems. As a result, the resulting heat flux level obtained by the empirical model is much smaller than that obtained by CFD. It implies that the carefull selection of the reference length depending on the gemetry type and the flow directions should be considered to achieve high-order accurate heat flux prediction.

5 CONCLUSION

Accuracy improvement of the destructive re-entry risk analysis methodology is pursued in this study to reduce the uncertainties on the expected casualty predictions and to realize the design-for-demise. Uncertainty quantification strategy was proposed, and the role of the high-fidelity numerical simulations was discussed. Based on investigation of the heat flux distributions and its magnitude level predicted by CFD for the flat faced disk and the rocket upper stage, the selection of the reference length to predict the stagnation heat flux, the considerations on the local curvature and the shock-interaction effect were found to be the key factors to achieve high-order accurate heat flux predictions.

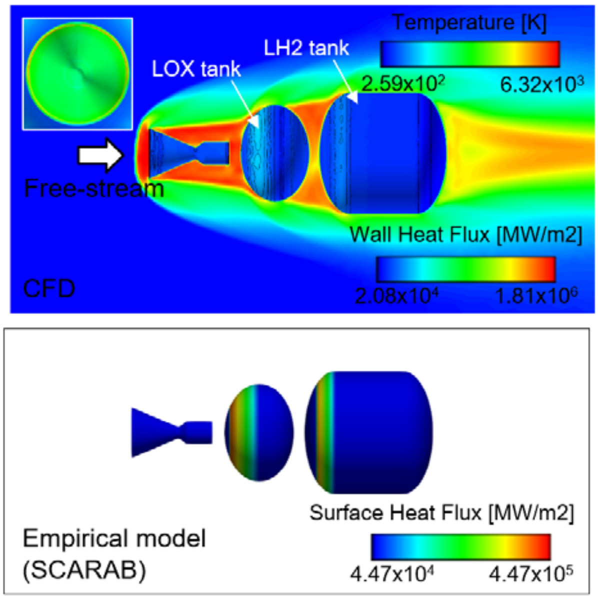

Fig. 6 Surface heat flux distributions evaluated by CFD and the empirical model [5] for pitch angle zero.

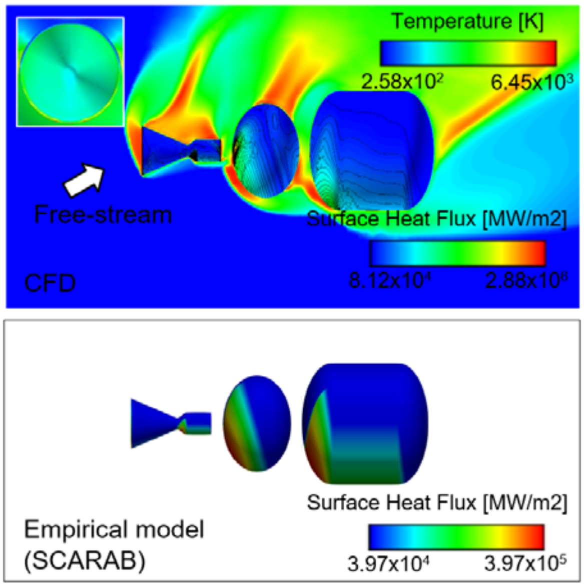

Fig. 7 Surface heat flux distributions evaluated by CFD and the empirical model [5] for pitch angle 30 degrees.

6 REFERENCES

1. Fujimoto, K., Tani, H., Negishi, H., Saito, Y., Iizuka, N., and Okita, K., "Update of Aerodynamics and Heat Flux Model for ORSAT-J", 8th IAASS Conference, 2016.

2. Fujimoto, K., Saito, Y., Shimizu, R., Kawamoto, S., and Ohnishi, M., "Overview and Progress of Research on Space Debris in JAXA", Stardust Final Conference on Asteroids and Space Debris, 2016.

3. Dobarco-Otero, J., et al., "Upgrades to Object Re-entry Survival Analysis Tool (ORSAT) for spacecraft and Launch Vehicle Upper Stage Applications", AAS Science and Technology Series, vol. 103, Space Debris 2000, Univelt, 2001, pp. 223-228.

4. Omaly P., Magnin Vella C., and Galera S., DEBRISK, "CNES tool for re-entry survivability prediction: validation and sensitivity analysis", Proceedings of the 6th IAASS Conference, Montréal (Ca), 21-23 May 2013.

5. T. Lips, B. Fritsche, M. Homeister, G. Koppenwallner, H. Klinkrad and M. Toussaint, "Re-entry Risk Assessment for Launchers – Development of the New SCARAB 3.1L", Proceedings of the 2nd IAASS Conference, 2007.

6. Annaloro, J. (2015) "Elaboration of New Spacecraft-orientated Tool: PAMPERO", 8th European Symposium on Aerothermodynamics for Space Vehicles, Lisbon, 2-6 March 2015.

7. Detra, R. W., Kemp, N. H. and Riddell, F. R., "Addendum to Heat Transfer to Satellite Vehicles Re-entering the Atmosphere," Jet Propulsion, vol. 27, no. 12, Dec. 1957, pp. 1256-1257.

8. G.Bird,"Molecular gas dynamics and the direct simulation of gas flows," Oxford Engineering Science Series, 42, 1994.

9. Tani, H., Ohmaru, T., Takata, S., Uematsu, H., Matsuura, M., Daimon, Y. Negishi, H., "Hybrid N-S/DSMC Simulations of HTV Main Engine Plume Impingement to the ISS, " Journal of the Japan Society for Simulation Technology, Vol. 34, No. 3, pp.186-192, 2015.

10. Koura, K. and Matsumoto, H., "Variable soft sphere molecular model for inverse-power-law or Lennard-Jones potential, " Phys. Fluids, A3 (1991), pp.2459-2465.

11. Bird, G. A., "Monte-Carlo simulation in an engineering context, Proceedings of the 12th International Symposium on Rarefied Gas Dynamics," Prog. Astronaut. Aeronaut., 74, 239, 1981.

12. Boyles, K.A., et al.,"The Use of Virtual Sub-Cells In DSMC Analysis of Orbiter Aero-dynamics at High Altitudes Upon Re-entry", AIAA-2003-1030, 41st AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan. 2003.

13. G.A. Bird, "Chemical Reactions in DSMC," 27th International Symposium on Rarefied Gas Dynamics, Pacific Grove, CA, July 10-15, 2010.

14. Fujimoto, K., "Study on the Automated CFD Analysis Tools for Conceptual Design of Space Transportation Vehicles," Ph.D. Dissertation, Univ. of Tokyo, Tokyo, Japan, 2006.

15. Kitamura, K., Nonaka, S., Kuzuu, K., Aono, J., Fujimoto, K., and Shima, E.,"Numerical and Experimental Investigations of Epsilon Launch Vehicle Aerodynamics at Mach 1.5," Journal of Spacecraft and Rockets, Vol.50, No.4, pp.896-916, 2013.

16. Negishi, H., Aono, J., Shimizu, T., Sunakawa, H., Sezaki, C., Nagao, N., Nanri, H., "Mixing Characteristics of Transcritical Hydrogen Flows around a Mixer in a Liquid Rocket Engine," JSASS-2016-2117-A, 2016 (in Japanese).

17. Kitamura, K., Fujimoto, K., Shima, E., Kuzuu, K., and Wang, Z. J., "Validation of Arbitrary Unstructured CFD Code for Aerodynamic Analyses," Transactions of the Japan Society for Aeronautical and Space Sciences, Vol.53, No.182, pp.311–319, 2011.

18. Fujimoto, K., Nambu, T., Negishi, H., and Watanabe, Y., "Validation of LS-FLOW for Re-entry Capsule Unsteady Aerodynamic Analysis", AIAA Paper 2017-1411, 55th AIAA Aerospace Sciences Meeting, AIAA SciTech Forum, 2017.

19. Haga, T., and Kawai, S., "Toward accurate simulation of shockwave-turbulence inter-action on unstructured meshes: A coupling of high-order fr and lad schemes," AIAA Paper 2013-3065, 2013.

20. Fujimoto, K., Fujii, K., and Wang, Z. J., "Improvements in the Reliability and Efficiency of Body-Fitted Cartesian Grid Method," AIAA Paper 2009-1173, 2009.