Numerical Analysis of Supersonic Jet Flow from Vertical Landing Rocket Vehicle in Landing Phase

Toshiyuki Suzuki*, **Satoshi Nonaka**† **and Yoshifumi Inatani**†

*Institute of Aerospace Technology
Japan Aerospace Exploration Agency (JAXA)
7-44-1, Jindaiji-Higashi-machi, Chofu-shi, 182-8522, Tokyo
E-mail: suzuki.toshiyuki@jaxa.jp - Web page: http://www.jaxa.jp

†Institute of Space and Astronautical Science
Japan Aerospace Exploration Agency (JAXA)
3-1-1 Yoshinodai, Sagamihara-shi, 229-8510, Kanagawa

Key Words: Opposing jet, Large-eddy simulation, Domain decomposition and MPI

1. INTRODUCTION

As a future space transportation system, a working group at Institute of Space and Astronautical Science (ISAS) of Japan Aerospace Exploration Agency (JAXA) proposed a concept of a fully reusable launch vehicle (RLV) [1]. The proposed RLV is a single stage vertical taking off and landing (VTOL) type vehicle. For the purpose of the verification of this concept, a small-scale test vehicle shown in Fig. 1 was built, and flight tests have been conducted. One of the most considerable issues in respect to the aerodynamics of the RLV is the landing control using its engine thrust. In the landing phase, the RLV ejects supersonic jet toward subsonic freestream so as to obtain enough drag force for vertical landing. Because the flowfield is significantly disturbed by the jet ejection resulting in highly unsteadiness characterized by massive separation and presence of vorticies, the aerodynamic forces acting on the RLV is greatly affected by the disturbance. Therefore, it is important to understand the unsteadiness of flowfield for conducting attitude control, guidance and navigation while descending. For this purpose, several wind tunnel experiments have been carried out in JAXA [2, 3].

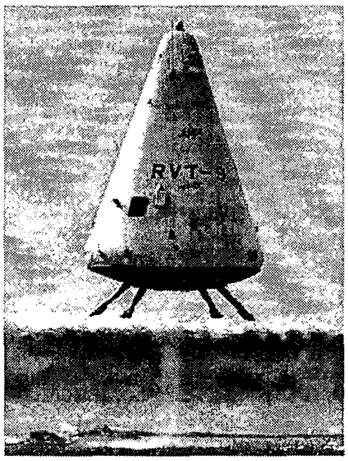


Fig. 1 Flight test of small test vehicle (Oct. 2003)

In the recent work of Nonaka et al., a 1/12-scale model of the test vehicle which can produce an opposing supersonic jet was built and a subsonic wind tunnel testing was made using the model to experimentally simulate the jet/freestream interaction. The flowfield around the model was visualized using the particle image velocimetry (PIV) technique[3]. From the instantaneous velocity vectors deduced from the image, it was found that the opposing jet from the model produced a vortex ring around the vehicle, and that the vortex ring moved unsteadily in another moment. Due to this unsteady flow motion, aerodynamic forces measured on the model were fluctuated with time. Unfortunately, because the wind tunnel experiment could not reproduce some parameters in the flight environment of the test vehicle such as Reynolds number and an engine nozzle exit diameter, etc., it is uncertain whether the observed phenomena in the experiment would occur in a flight environment. Therefore, a computer code is needed to be able to simulate the jet/freestream interaction accurately. Such a computer code will be helpful to correlate the experimental data to a realistic flight environment of the small-scale test vehicle, and of a full scale RLV in the future.

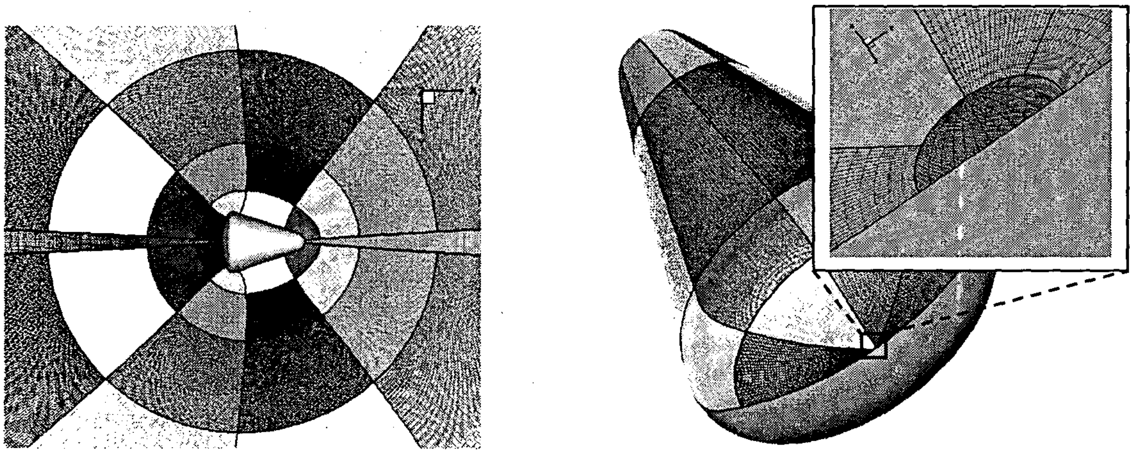
Large-eddy simulation (LES) is expected to be a powerful and efficient technique to analyze such an unsteady flowfield. However, LES itself requires a fine grid to resolve the small scales of turbulence in the high Reynolds number jet flow. Moreover, a calculation of the jet/freestream interaction flowfield needs to treat a large scale of computational domain that is extended to farfield of an external flow. Because a considerable number of grid points is required for this kind of computation, a parallel computation teqnique is indispensable to obtain solutions in reasonable CPU time.

The ultimate aim of the present study is to simulate the jet/freestream interaction flowfield around the vehicle in order to examine how the qualitative nature of the flowfield affects the aerodynamic forces on the RLV. For this purpose, we calculate the flow environment for the wind tunnel conditions by Nonaka et al [3] using LES with domein decomposition parallerization teqnique in this paper. Calculated results are compared with the experimental data to validate our LES modeling. Simulation of the actual flight environment of the small-scale test vehicle will be studied in future work.

2. NUMERICAL METHODS

Configuration of Test Model and Computational Grid

Figures 2(a) and (b) show a typical example of block-structured computational grid around the vehicle. Note that only half of physical domain is used for computation because of symmetry. The grid is designed to provide an adequate resolution of the dominant mean flow structures near the jet/freestream interaction region, and contains 14.1 million points distributed over 66 blocks. There is a nozzle at the center of the base region. The diameter of the nozzle exit is 5% of the base diameter. As shown in Fig. 2(b), the grid near the nozzle exit is modified to remove the singularity at the stagnation point and to reduce the number of grid points. The computational grid uses viscous grid spacing suitable for turbulent boundary layer computations at body surface.



(a) Cross section of computational mesh containg 66 blocks

(b) Surface mesh and close up view around nozzle exit region

Fig. 2 Computational mesh used for present LES analysis.

Computational Fluid Dynamics Solver

Numerical methods used in the calculation are briefly described. In this study, we use a flow solver called Unified Platform for Aerospace Computational Simulation (UPACS)[4], a standard CFD code in Institute of Aerospace Technology (IAT) of JAXA. This collaborative computational software is designed to be shared among researchers. Besides flexibility and extendibility, reliability is specially emphasized in its development. The code is parallelized by a flexible domain decomposition concept and MPI. The computations are accomplished using 11 to 66 processors of Fujitsu PRIMEPOWER HPC2500, which is the central machine of Numerical Simulator III system in JAXA. The Navier-Stokes flow solver of the current version is based on a cell-centered finite-volume method on multi-block structured grids.

Large eddy simulation is performed in order to capture the unsteady fluctuations accurately.

Smagorinsky constant is set to Cs=0.1. The convection terms are discretized utilizing AUSM-DV scheme and MUSCL approach for for attaining 2nd-order spatial accuracy. The viscous terms are discretized using 2nd-order central scheme. In order to avoid numerical instability, 6th-order filter developed by Lele is used with filter constant of α=0.45. Time integration is performed by Matrix-Free Gauss-Seidel (MFGS) scheme with 3 sub-iterations. The time step is set to dt=1.0×10^{-4} in order to obtain power spectral density of the pressure coefficient fluctuations in reasonable CPU time.

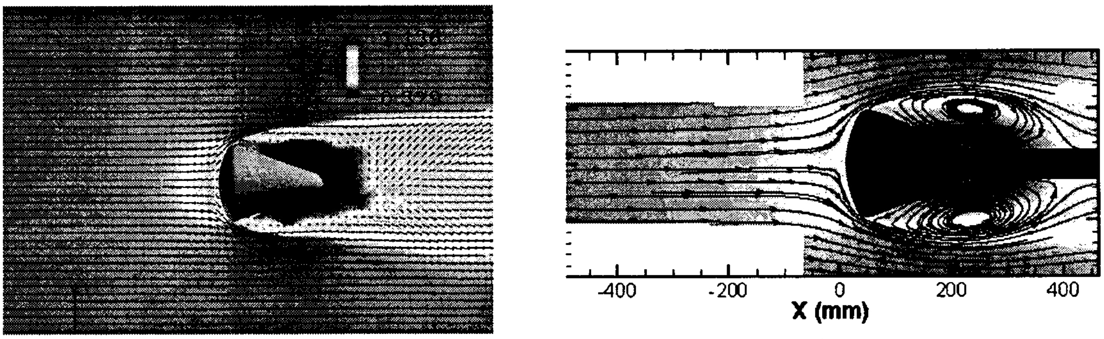
3. RESULTS AND DISCUSSIONS

Table 1 Freestream condition and nozzle exit condition of jet ejection

Freestream	
Pressure [Pa]	101330
Density [kg/m3]	1.205
Temperature [K]	291.3
Velocity [m/s]	26.4
Mach number	0.077
Reynolds number	323000
Nozzle exit	
Pressure ratio	1.38
Mach number	2.41

LES of unsteady flowfield under wind tunnel test condition is carried out. In this study, we assume the angle of attack to be 0 deg. The freestream condition and nozzle exit condition of opposing jet in the wind tunnel experiment are summarized in Table 1. These conditions are corresponded to the flight condition when the small test vehicle descends at 70 m/s and throttles at 100 % thrust.

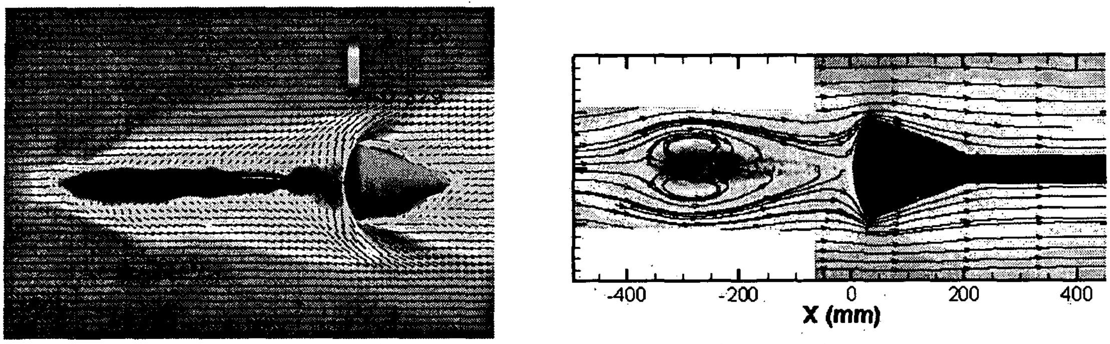
Figure 3 and 4 show the velocity components obtained by the LES at cross sectional plane of symmtry (z/c=0) for the case without and with jet ejection, respectively. Experimental results obtained by using PIV technique are also shown for the purpose of comparison. The vectors indicate the two components of velocity on the plane and the color contour describes the velocity component along freestream direction. The velocity is normalized by the freestream velocity, and the same scale is used for all of plotting. Since LES results are unsteady, time-averaged data are used. As shown in Fig. 3, the velocity at the vehicle base region becomes lower because of the stagnation flow. Besides, a flow separated around the courner of the vehicle makes a large recirculation region at the side of the vehicle. These trends correspond with those of measured by the PIV experiment. In the case with jet ejection, on the other hand, a stagnation point is formaed ahead of the jet flow. In addition, flow separation that is observed in the case without jet ejection can not be recognized, and the flow is attached to the vehicle. Though a qualitative agreement between the LES and PIV is fairly good, a jet flow region (designated blue in Fig. 4) that calculated by the LES is slightly larger than that of measured. The reason for this difference is unknown. This point needs to be clarified in the future.



(a) LES (time-averaged) (b) Experiment (PIV)

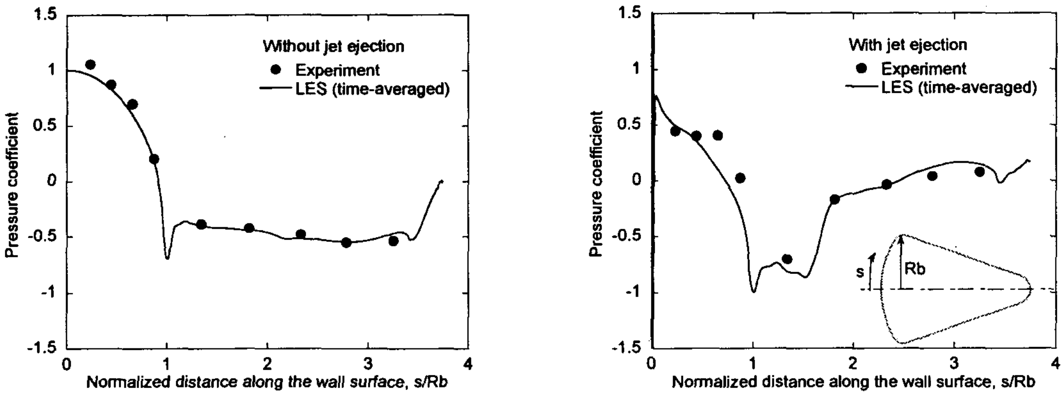
Fig. 3 Velocity field on cross sectional plane of symmetry for the case without jet ejection

The pressure coefficient profile along the entire surface is compared in Figs. 5(a) and (b) for the case without and with jet ejection, respectively. In both the LES and experiment, the pressure coefficient at the base region for the case with jet ejection is lower than the case without jet ejection. This is because the freestream toward the base is biased by the jet flow region, and goes further downstream rather smoothly. As a result, a dynamic pressure acting on the vehicle base is reduced. On the other hand, the pressure along the side surface for the case with jet ejection is higher than the case without jet ejection. This is becase the flow biased by the jet ejection is attached to the side surface of the vehicle as shown in Fig. 4. Consequently, pressure becomes higher along the side surface. Note that calculated pressure coefficients along the entire body surface for both cases duplicate well with those obtained by experiment.



(a) LES (time-averaged) (b) Experiment (PIV)

Fig. 4 Velocity field on cross sectional plane of symmetry for the case with jet ejection



(a) Without jet ejection (b) With jet ejection

Fig. 5 Comparison of pressure coefficient between LES and measurement

4. CONCLUSION

In order to study the effect of the jet ejection from the RLV while descending, flowfield analysis under wind tunnel test condition is performed using LES technique. The time-averaged components of the flowfield are compared with those obtained by the experiment. In the full paper, we will introduce the unsteady component of the flow features.

REFERENCES

[1] Inatani, Y., Naruo, Y. and Yonemoto, K., “Concept and Preliminary Flight Testing of a Fully Reusable Rocket Vehicle,” *Journal of Spacecraft and Rockets*, Vol. 38, No. 1, 2001, pp. 36-42.

[2] Akita, D., “Aerodynamic Effects of Opposing Nozzle Jet on Reusable Vertical Landing Rocket,” Proceeding of 23rd International Symposium on Space Technology and Science, ISTS-2002-s-08, 2003, pp. 2418-2423.

[3] Nonaka, S., Osako, Y., Ogawa, H. and Inatani, Y., “Aerodynamics of Vertical Landing Rocket Vehicle in Landing Phase,” Advances in the Astronautical Sciences, Vol. 117, pp. 791-803, 2004.

[4] Yamane, T., Yamamoto, K., Enomoto, S., Yamazaki, H., Takaki, R. and Iwamiya, T, “Development of a Common CFD Platform - UPACS -,” Proceedings of the Parallel CFD 2000 Conference, Trondheim, Norway, Elsevier Science B. V., 2001, pp. 257-264.