Coupled Simulation of Shock Waves in Gas-Particle Mixtures Introducing Motion Equations

Yusuke MIZUNO*, Shun TAKAHASHI*, Taku NONOMURA† , Takayuki NAGATA* and Kota FUKUDA*

* Tokai Univesity, 4-1-1 Kitakaname, Hiratsuka, Kanagawa, JAPAN,
e-mail: 1beu2104@tokai-u.jp

* Tokai Univesity, 4-1-1 Kitakaname, Hiratsuka, Kanagawa, JAPAN,
e-mail: takahasi@tokai-u.jp

† ISAS/JAXA, 3-1-1 Yoshinodai, Chuo-ku, Sagamihara, Kanagawa, JAPAN,
e-mail: nonomura@flab.isas.jaxa.jp

* Tokai Univesity, 4-1-1 Kitakaname, Hiratsuka, Kanagawa, JAPAN,
e-mail: 1beu2216@tokai-u.jp

* Tokai University, 4-1-1 Kitakaname, Hiratsuka, Kanagawa, JAPAN,
e-mail: fukuda@tokai-u.jp

ABSTRACT

The acoustic waves generated from rocket plumes are sufficiently strong to damage the satellites inside the fairing of a rocket. Fukuda et al. showed the acoustic wave might be primarily attenuated by interactions between particles and turbulence[1). These particles are alumina particles exhausting from the rocket nozzle or water drops surrounding the rocket. The aim of this work is validation of a phenomenon of shock waves in gas-particle mixtures using direct numerical simulation (DNS). A flow solver based on three-dimensional compressible Navier-Stokes equations is developed for the purpose of high accurate prediction of the acoustic field around a rocket. A switching-type numerical scheme is employed in the flow solver to capture both turbulence and shock regions accurately by the second-order pseudo skew-symmetric scheme and the monotone upstream-centered scheme for conservation laws (MUSCL) scheme. This flow solver is capable of analysing a flow around moving multiple particles in the three-dimensional flow field[2). The flow around moving multiple particles and interference with the shock wave at high Mach and low Reynolds number is resolved and compared with results obtained from a high-order boundary-fitted-coordinate (BFC) solver. We investigated the flow around moving particles of number of up to 16, introducing motion equations and collision detection. Also different models of the drag coefficient, affecting the interaction between the gas and particles, the results are compared. The Mach and Reynolds numbers are set to be 0.3-2.0 and 50-300, respectively. The pressure distribution around two particles just before the frontal collision is shown in Fig. 1. The pressure distribution of multiple (16) moving particles of the is shown in Fig. 2.

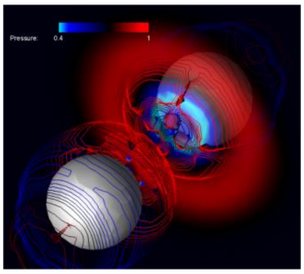

Fig. 1 The frontal collisio

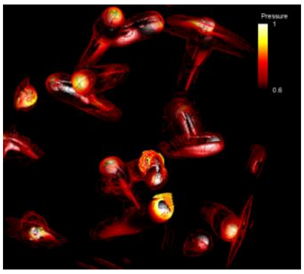

Fig. 2 Multiple moving particles

REFERENCES

[1] K. Fukuda, S. Tsutsumi, T. Shimizu, R. Takaki, and K. Ui, "Examination of Sound Suppression by Water Injection at Lift-off of Launch Vehicles", The 17th AIAA/CEAS Aeroacoustics Conference, AIAA paper 2011-2814, 2011.

[2] S. Takahashi, T. Nonomura and K. Fukuda, ''A Numerical Scheme Based on an Immersed Boundary Method for Compressible Turbulent Flows with Shocks: Application to Two-Dimensional Flows around Cylinders'', *Journal of Applied Mathematics*, vol. 2014