Proceedings, ICCES, vol.23, no.1

Multi-physics CFD Simulation in a Jet Engine

Makoto Yamamoto[1*]

[1]Tokyo University of Science, 6-3-1 Niijuku, Katsushika-ku, Tokyo 125-8585, Japan
[*]Corresponding Author: Makoto Yamamoto. Email: yamamoto@rs.tus.ac.jp

Abstract: In a turbine of a jet engine, deposition phenomenon is often observed. Deposition is a phenomenon that particles such as volcanic ash, sand and dust passing through a combustion chamber of a jet engine are melt, rapidly cooled and then accumulate on the turbine blade and end-wall surfaces. Deposition is one of critical problems when aircraft flies in a cloud with many particles. Obviously, deposition can degrade the aerodynamic performance of the turbine blade and vane, and make partial or complete blockage of film-cooling holes. As the result, deposition deteriorates safety and life time of the turbine. In the past investigations, a lot of researches have focused on the effect of deposition on aerodynamic performance and cooling efficiency. The dependency of deposition on wall temperature was also investigated. However, the deposition mechanism itself remains unclear, especially at high temperature condition in a jet engine. This is because it is too difficult to conduct the detailed measurements due to the small size of particles and the high temperature in a turbine. Additionally, as it can be expected that deposition is a typical multi-physics phenomenon, the physics is very complex. Therefore, a reliable deposition model has not been established and validated yet. If the mechanism could be clarified, it is possible to develop a good deposition model and realize more accurate predictions of deposition phenomenon in a jet engine. Considering these backgrounds, in the present presentation, I will explain our computational researches on deposition phenomenon in a jet engine. First, a numerical method to predict deposition phenomenon in a jet engine is proposed, based on the grid-based method and multi-physics modeling. Some existing deposition models which were empirically derived are validated to a canonical condition, and then the best model is applied to a turbine vane. Through this study, the characteristics of deposition on the turbine vane is clarified. Second, in order to clarify the deposition mechanism, solidification of a high temperature droplet is reproduced, using the particle-based method. Based on the numerical results, the detail of solidification process is investigated.

Keywords: Computational fluid dynamics; multi-physics modeling; deposition phenomenon; high temperature; turbine vane; jet engine

DOI: 10.32604/icces.2021.08478