Paper ID: 78160
student

IAF SPACE PROPULSION SYMPOSIUM (C4)
Hypersonic Air-breathing and Combined Cycle Propulsion, and Hypersonic Vehicle (7)

Author: Mr. Kotaro Iguchi
Kanazawa Institute of Technology, Japan, c6300294@st.kanazawa-it.ac.jp

Mr. Masanori Harada
Kanazawa Institute of Technology, Japan, c6200307@st.kanazawa-it.ac.jp
Prof. Hideki Moriai
Kanazawa Institute of Technology, Japan, moriarty@neptune.kanazawa-it.ac.jp

A STUDY ABOUT THE SIMPLE NUMERICAL SIMULATION METHODS TO ANALYZE THE FLAME HOLDER FLOW FIELD OF THE HYPERSONIC JET ENGINES

Abstract

Hypersonic engines are expected to be used for future space travel and intercontinental high-speed transportation. However, the development of hypersonic engines with high enthalpy flow requires large-scale test facilities, a large number of hardware fabrication and testing trials, namely the enormous cost, manpower, and time for these processes. Therefore, the application of CFD analysis and other numerical simulation techniques is important to promote development with fewer development resources. However, CFD analysis of such hypersonic high-enthalpy flows is technically difficult, and the current prediction accuracy is not always sufficient. In addition, since the number of meshes in a 3D simulation is in the order of a billion, even using a supercomputer, a huge amount of computation time of several weeks or more per case is required, making the application to multi-objective optimization needed for speedy hardware design and development rather difficult at the present time. Therefore, the purpose of this research is to develop simple simulation methods that can perform supersonic combustion flow field calculations on a workstation, and to confirm the feasibility of multi-objective optimization. Since flame holding inside a hypersonic engine is a difficult and important technique, this research focuses on flame holders of the hypersonic engines. In order to accommodate various flame holder flow fields, two types of hypersonic engines were selected for simulation trial: A Rocket-Based Combined Cycle (RBCC) engine and a scramjet engine. For the results of the hypersonic wind tunnel tests and full 3D LES analyses, a computationally less demanding analysis methods combining an extremely thin 3D geometry model and a simple one-step chemical reaction model was applied. In addition, various flow modeling methods were compared, including RANS and LES, steady and unsteady, and initial turbulence and periodic boundary conditions in the depth direction. As a result, the combination of the appropriate modeling methods was able to qualitatively reproduce the pressure distribution of the tests in the RBCC engine and the mixing behavior of full 3D LES inside the flame holder of the scramjet engine. These results indicate that it is not unrealistic to analyze many such complex flow field on a workstation for multi-objective optimization by applying the obtained simple simulation methods. Finally, the obtained analysis methods were applied to optimize the flame holder geometry and fuel injection design in previous research, and the improvement in mixing and flame holding performance was confirmed.