Paper ID: 43126
student

IAF SPACE PROPULSION SYMPOSIUM (C4)
Hypersonic Air-breathing and Combined Cycle Propulsion (9)

Author: Dr. Tingting Jing
Northwestern Polytechnical University, China

Dr. Fei Qin
China
Mr. Xianggeng Wei
Northwestern Polytechnical University, China

FLOW AND THERMAL CHARACTERISTICS IN REGENERATIVE COOLING CHANNELS AROUND CAVITY OF RBCC

Abstract

As one of the most prospective approaches to realize reusable orbit transportation system and near space hypersonic vehicle, Rocket Based Combined Cycle (RBCC) engine combines the air-breathing cycles and a rocket subsystem to obtain high thrust-to-weight ratio and high specific impulse. In order to realize high combustion efficiency, various of flame holding techniques such as cavity and pylon are utilized in the flow path of RBCC. However, the flame holder structure will influence the distributions of flow and combustion parameters, such as velocity, temperature, heat transfer coefficient, etc., accordingly, the thermal protection structure should also change with the specific structure. In the present paper, the flow and thermal behaviors in the regenerative cooling channels around cavity were investigated with a 3D coupled numerical model. Firstly, the characteristics of thermal environment around the cavity was analyzed. The simulation result shows that the expansion around the leading edge of cavity and convergence around the trailing edge form a low velocity zone followed with an accelerating zone, therefore, the heat flux on the cavity's wall changes in the axial direction. However, the flow field in the regenerative cooling channel changes in both axial and radial directions because of the Dean vortices and centrifugal force around the corner. The combining effects of the inner thermal environment and outer cooling behavior differ the thermal protection structure around the cavity with other components. Secondly, the influence of coolant temperature was investigated. The pressure in the cooling channel is always above the critical point of the fuel, the thermal physical parameters exhibit dramatic changes which will magnify the complexities of flow and heat transfer processes in the cooling channel of cavity. And the inlet temperature also corresponds to different species because of the pyrolysis reaction, therefore, the effect of inlet temperature is also investigated. Thirdly, an optimized thermal protection structure considering the coupled influences of inner and outer characteristics around cavity is proposed, the simulation result shows that the optimized structure could meet the requirement of thermal protection around cavity.