IAF SPACE PROPULSION SYMPOSIUM (C4)
Interactive Presentations - IAF SPACE PROPULSION SYMPOSIUM (IP)

Author: Mr. ANANT SINGHAL
Indian Space Research Organization (ISRO), Liquid Propulsion Systems Centre (LPSC), India, mail2anantsinghal@gmail.com

Mr. Deepak K. Agarwal
Indian Space Research Organization (ISRO), India, dagarwal_iitk@yahoo.com
Dr. John T Tharakan
Indian Space Research Organization (ISRO), Liquid Propulsion Systems Centre (LPSC), India, tharakan12@gmail.com
Dr. Sunil Kumar S.
Indian Space Research Organization (ISRO), India, ted.prsg@gmail.com

NUMERICAL SIMULATION OF HIGH TEMPERATURE GAS CONDENSATION IN A FLOWING CRYOGENIC LIQUID IN SEMICRYOGENIC ENGINE

Abstract

In the current study condensation of high-temperature gaseous oxygen on mixing with subcooled liquid oxygen flowing in a curved pipe is simulated using CFD. Gaseous oxygen is injected into the flowing liquid oxygen through circumferential holes on the pipe. Such a scenario of two-phase flow is encountered in the booster pump exit line of liquid oxygen-kerosene rocket engine. The turbine drive hot gas, which is mostly gaseous oxygen is mixed with liquid oxygen at the exit of booster pump and subsequently condenses downstream. Gaseous oxygen has to be fully condensed before reaching main pump inlet to avoid performance degradation of main pump. Though CFD simulations on steam injection into water are reported in literature, the detailed study with cryogenic fluid is absent. The present study determines the two-phase flow behaviour and heat transfer characteristics of the gaseous oxygen mixing with liquid oxygen for various configurations and momentum ratios. A commercial CFD software, ANSYS Fluent, is used for the present analysis. It consists of a curved pipeline having circumferential inlets for injection of gaseous oxygen.The gas-liquid two-phase flow is formulated in the Eulerian-Eulerian framework. Liquid oxygen is in continuous phase while gaseous oxygen is in dispersed phase. Thermal Phase Change (TPC) model is used to simulate interaction between the two phases. The TPC model considers heat transfer on each side of the phase interface using two-resistance method. Mass transfer across the gas-liquid interface is subsequently calculated from the heat balance. The effect of fluid parameters viz. gas inlet temperature and gas mass flow rates, is investigated. Various two-equation RANS turbulence models are used to study its impact on the flow field. The study is also carried out to determine the impact of gas bubble diameter on the condensation process and is found to be an important parameter for the phase change. The study shows that the use of different turbulence models does not significantly affect the amount of gas condensed. However, the effect of variation of the fluid parameters on the gas condensation is found to be significant.