IAF SPACE PROPULSION SYMPOSIUM (C4)
Propulsion Technology (1) (3)

Author: Mr. KHALID RASHID
Indian Space Research Organization (ISRO), India, khalid9835@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 INVESTIGATION OF ROTATING STALL IN HIGH PRESSURE FUEL PUMP OF SEMICRYOGENIC ENGINE

Abstract

A staged combustion cycle based high thrust Semicryo engine of vacuum thrust of 2000kN is under development. Propellants namely liquid oxygen and Isrosene are fed into two separate combustion chambers called preburner and thrust chamber. In order to achieve high chamber pressure for its operation, propellants are pressurised using centrifugal pumps. The high pressure fuel pump (HPFP) is two stage pump which consists of Main fuel pump(MFP) for supplying fuel to thrust chamber and Preburner pump(PBP) to handle required Isrosene for the pre-burner. These pumps can exhibit different kinds of unsteady phenomena depending on type of interaction and flow behavior changes happening inside the pump which leads to pressure fluctuation at pump delivery section. As this is a hydrocarbon-based engine, this will lead to combustion instabilities, which are expressed as unsteady large-amplitude pressure fluctuation in the thrust chamber and may further excite multi-frequency oscillations inside the chamber. Thus, evaluation of different design parameters of centrifugal pump is vital in minimizing these pressure oscillations. One major concern in centrifugal pump is the presence of rotating stall which is an unsteady flow phenomenon that causes instabilities and low frequency oscillation in pumps. In view of this transient three-dimensional CFD analysis of high-pressure fuel pump is carried out to determine its flow characteristics under rotating stall and its unsteady behaviour inside the centrifugal pump under different flow rates are analysed. CFD results are validated with cold flow test carried out with water and oscillation frequency and amplitude are in good agreement with the test data. Further effect of impeller diameter, impeller exit width and diffuser setting angle on stall behaviour, head rise and pressure fluctuations are determined. Subsequently performance is evaluated for the throttling range varying from 70% to 110% of nominal thrust.