IAF SPACE PROPULSION SYMPOSIUM (C4)
Liquid Propulsion (1) (1)

Author: Mr. Abhishek Sharma
Indian Space Research Organization (ISRO), India, abhisheksharma@lpsc.gov.in

Dr. John T Tharakan
Indian Space Research Organization (ISRO), Liquid Propulsion Systems Centre (LPSC), India, tharakan12@gmail.com

ANALYSIS FOR DESIGN OPTIMIZATION OF HIGH THRUST LIQUID ENGINE HOT TEST FACILITY

Abstract

A liquid oxygen-kerosene based high thrust engine is being developed for future launch vehicle applications. It is based on staged combustion cycle, with engine operating at nominal chamber pressure of 180bar. The engine will undergo hot tests under sea level conditions for flight qualification and acceptance of launch vehicle stage. The engine and stage will be tested in vertical mode and rocket plume is sufficiently cooled before impinging on facility walls to avoid any hazard to engine and adjoining mechanical structures. The LOX-kerosene engine will be mounted on loading frame above ground level and water will be directly injected into the rocket plume. An adequate quantity of water must be injected to maintain the facility wall temperature within safe limits. An extensive design optimization study is carried out to ascertain safe operating conditions of hot test facility. A validated methodology based on simulation of existing test stand is developed initially and was employed to perform two phase Computational Fluid Dynamics (CFD) analysis for design of LOX-kerosene test facility. A detailed numerical study to test major design parameter like water injection configuration was performed. A discrete particle (DPM) based two phase methodology was used to model water droplets. A RANS based Eulerian-Lagrangian two phase model was developed in finite volume framework. SST k-omega model with compressibility correction was invoked to take care of diffusion dissipation in high Mach number flows. Analysis of initial design configuration highlighted the inefficiency of water injection configuration, which resulted in higher facility wall temperature above safe working limit. To extract maximum advantage of available water flow rate at 60m/s injection velocity, a multi-plane injection scheme was devised to appropriately apportion coolant flow at two axial locations of rocket plume. A detailed parametric CFD study was then carried out to determine thermal condition of facility for various multi-plane injection sequences. Multiple simulations were carried to determine water injection pattern at two planes which can efficiently cool the rocket plume and provide lower adiabatic wall temperature. Analysis displayed the efficacy of dense concentrated water jets in multi-plane injection configuration. A substantial reduction from 2800K (initial design configuration) to 1150K with the newly devised multi-plane injection was observed. The study highlights the role and development of CFD based methodology, which is used to determine an appropriate water injection configuration for design of LOX-kerosene engine hot test facility. A new water injection configuration is devised based on numerous computational trials, which can be suitably implemented at engine test stand.