IAF SPACE PROPULSION SYMPOSIUM (C4)
Propulsion Technology (3) (10)

Author: Mr. Ajith M
Vikram Sarabhai Space Centre, ISRO, Thiruvananthapuram, India, ajithtkmce@gmail.com

Mr. Dileep KN
Vikram Sarabhai Space Centre, ISRO, Thiruvananthapuram, India, kn_dileep@vssc.gov.in
Mr. Levin G
Indian Space Research Organization (ISRO), India, g_levin@vssc.gov.in
Mr. Lazar Chitilappilly T.
Vikram Sarabhai Space Centre (VSSC), India, lt_chitilappilly@vssc.gov.in
Mr. Eswaran V
Indian Space Research Organization (ISRO), India, v_eswaran@vssc.gov.in

CFD SIMULATION OF REACTIVE FLOW IN CRYOGENIC ROCKET NOZZLE AND PERFORMANCE PREDICTION OF LVM III CE20 NOZZLE

Abstract

Accurate prediction of rocket-nozzle performance is vital in rocket engine development and selection of engine operating systems. Numerical simulation is a cost effective alternative for various parametric studies to optimize rocket nozzle. The chemical and physical evolution of a complex multi species reactive flow system in rocket nozzle can be numerically simulated using computational fluid dynamics. A reacting flow field can be simulated by adding finite rate chemistry to standard compressible Navier-Stokes equations with proper turbulence model, thermodynamic model and reaction kinetics. The present work describes the development of numerical methodology to simulate the reactive flow within the cryogenic engine nozzle and prediction of nozzle performance. Six species eight step reaction chemistry by JS Evans and CJ Schexnayder is used for modeling hydrogen-oxygen reaction within the nozzle. Species concentration in the combustion products which proceeds to the nozzle inlet are assumed based on chemical equilibrium analysis code. This species equilibrium composition was given at the inlet of the nozzle and ambient pressure at exit. This reactive flow simulation calculates the specific impulse of the nozzle after accounting for the boundary layer loss, kinetic loss, and divergence loss. Energy release loss is accounted by multiplying the predicted Isp value with C* efficiency obtained from experiment. The nozzle performance parameters predicted using this methodology was compared against the test data obtained during the experimental program conducted at altitude test capsule at NASA Lewis Rocket Engine Test Facility (RETF), where a contoured nozzle of area ratio (AR) 1030 and 427.5 was test fired with hydrogen-oxygen propellants at various mixture ratios. The present reacting flow solver was able to successfully simulate nozzle flow field and could predict the delivered specific impulse within the accuracy range of 0.1-1.82% of the experimental results. CFD results were compared with the Isp values obtained from JANNAF performance prediction procedure which uses various reference programs like ODE,ODK, MOC and TDK. The validated method was used for predicting the Isp value for the CE20 nozzle which powers the final cryogenic stage of ISRO's LVM III. Detailed CFD methodology, reaction chemistry and the comparisons with the test data for 1030AR as well CE20 nozzles are presented in the final paper.