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

Author: Mr. Krishnajith Jayamani
Indian Space Research Organization (ISRO), Liquid Propulsion Systems Centre (LPSC), India, krishnajith@lpsc.gov.in

Mr. Vasudevan R
Indian Space Research Organization (ISRO), Liquid Propulsion Systems Centre (LPSC), India, r_vasudevan@lpsc.gov.in
Mr. A.K. Asraff
Indian Space Research Organization (ISRO), India, akasraff@yahoo.com
Mr. Benoy Joseph
Indian Space Research Organization (ISRO), Liquid Propulsion Systems Centre (LPSC), India, k_benoyjoseph@lpsc.gov.in
Mr. Sajeesh N V
Indian Space Research Organization (ISRO), Liquid Propulsion Systems Centre (LPSC), India, nv_sajeesh@lpsc.gov.in
Mr. Sarath Chandran Nair S
Indian Space Research Organization (ISRO), Liquid Propulsion Systems Centre (LPSC), India, sarathcnairs@gmail.com

FINITE ELEMENT METHOD BASED FAILURE PREDICTION AND REDESIGN OF A LAUNCH VEHICLE VALVE

Abstract

The fluid feed circuit of liquid propellant based rocket engine employs several valves for directing and controlling the flow of fuel and oxidiser. The valves undergo several types of tests as part of their qualification process. One such test is the cyclic actuation test wherein it will be subjected to multiple numbers of actuations with or without the presence of actual pressurisation medium. During the course of such a test, cracks were observed in a newly designed valve of ISRO. The cracks were visually seen at an area that had only minor functional implication on the overall structural integrity of the valve. Nevertheless, numerical simulation of the valve using finite element method was performed to learn about the likely cause of these cracks. The analysis indicated that the observed cracks could be due to a combination of high cycle fatigue and low cycle fatigue. Furthermore, analysis predicted that the loading could cause even more severe damage to some of the internal components which if unmitigated, would impair the functionality of the valve and compromise its structural integrity.

The predictions from finite element simulations were confirmed by Non-Destructuve Testing of the valve which pointed towards permanent deformations of some of the internal components. In the light of these observations, the valve internals were redesigned so that the stresses and strains in it are well within their permissible limits. The structural adequacy of the re-designed configuration was established through finite element analysis. A qualification hardware was also fabricated and successfully subjected to the required number of cyclic actuations.

This paper presents the studies undertaken to establish the likely cause of the observed cracks in the original configuration of the valve. The failure mitigation by means and redesign, its finite element simulation and subsequent experimental validation is also presented.