Paper ID: 68732
student

50th STUDENT CONFERENCE (E2)
Student Conference - Part 1 (1)

Author: Mr. Adriano Casablanca
Delft University of Technology, The Netherlands, adriano.waitze@gmail.com

Dr. Angelo Cervone
Delft University of Technology (TU Delft), The Netherlands, a.cervone@tudelft.nl

PERFORMANCE INVESTIGATION OF VAPORIZING LIQUID MICRO-RESISTOJETS AND LOW-PRESSURE MICRO-RESISTOJETS FOR THE LUMIO MISSION

Abstract

The requirements of small satellite missions, especially interplanetary and deep-space ones, often generate a need for an RCS propulsion system. This happens in the Lunar Meteoroid Impact Observer (LUMIO), an upcoming CubeSat mission designed to observe, quantify, and characterize meteoroid impacts on the lunar far-side, in which Delft University of Technology is involved as a member of an international consortium of industrial and academic partners. A potential option for this RCS system are micro-resistojets, thanks to their low thrust, low mass, high scalability, high thrust-power ratio, and the use of non-hazardous "green" propellant. In this paper, the performance of Vaporizing Liquid Micro-Resistojets (VLM) and Low-Pressure Micro-Resistojets (LPM) will be analysed and compared using as a study case the mission requirements of the LUMIO spacecraft. A preliminary analytical model will be used to obtain an optimized geometry based on a trade-off from thrust, mass, volume, power, and compliance followed by a numerical simulation to confirm the results and better estimate the final performance of the propulsion system. The VLM is a classical resistojet that works by expanding vaporized propellant through a convergent-divergent nozzle; in this case, simulation of the gas flow will be done through the finite volume density-based OpenFOAM rhoCentralFoam solver. The LPM operates at a low pressure where the flow is in the transition regime, at a relatively high Knudsen number ($0.1 < \mathrm{Kn} < 10$), leading to a non-continuum flow which will be modelled through the Direct Simulation Monte Carlo (DSMC) method by the OpenFOAM dsmcFoam solver.