IAF MATERIALS AND STRUCTURES SYMPOSIUM (C2)
Space Vehicles – Mechanical/Robotic/Thermal/Fluidic Systems (7)

Author: Mr. Mayank Kumar
Vikram Sarabhai Space Centre, Thiruvananthapuram-695 022, INDIA, India, scimayank@gmail.com

REENTRY CONVECTIVE HEAT FLUX ESTIMATION USING CFD

Abstract

Indian Space Research Organization (ISRO) has announced its human spaceflight programme, which aims at human presence in a Low Earth Orbit for up to a week. The development of a human habitable spacecraft, capable of making controlled descent, is envisaged as well. Reentry heating is one of the most severe phase of such a manned spacecraft on its return back home. Detailed aerothermal characterization of such space vehicle is required so as to arrive at the appropriate heat flux required for the heat shield's thermal protection system design. To fulfill the same, high fidelity numerical simulations have been performed in order to ascertain the hypersonic aerothermodynamic flow field around a generic manned space capsule during reentry and compute the convective heat flux.

This paper details about the Computational Fluid Dynamics (CFD) simulations carried out to investigate the flow around a typical spherical nose-cone configuration reentry vehicle and predict the convective heat flux profile over the surface for a non-zero angle of attack. Grid generation and solver settings used to carry out the simulations have been discussed. Mesh dependency and turbulence closure sensitivity analysis have been carried out using axis-symmetric simulations and are reported as well.

The approach of predicting convective heat flux using CFD simulations is validated against the available flight and experimental heat flux data for similar class of reentry vehicles. The validation cases provide good confidence in the current study and help in firming the dispersion band on the heat flux values. Convective heat flux profiles at different Mach numbers for the reentry capsule is computed and briefly discussed. Overall flow features of the hypersonic reentry flow and its effect on the surface heat flux is analyzed as well. Effect of chemical equilibrium or non-equilibrium on the convective heat flux profile is assessed via chemical non-equilibrium CFD simulations. Wall catalyticity is found to be a major factor affecting heat flux, with non-catalytic wall heat flux being 50 percent less as compared to fully-catalytic wall heat flux.