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

Author: Mr. Sourabh Bhat
University of Petroleum and Energy Studies, India, spbhat@ddn.upes.ac.in

Dr. Ugur Guven
United States, drguven@live.com
Mr. Linsu Sebastian
University of Petroleum and Energy Studies, India, lsebastian@ddn.upes.ac.in
Mr. Karthik Sundarraj
University of Petroleum and Energy Studies, India, karthik_sundarraj@yahoo.com

PARAMETRIC SHAPE OPTIMIZATION OF REENTRY MODULE FOR SPACE MISSIONS

Abstract

Shape optimization for a reentry module is not trivial as it includes various factors such as the problem of reduction of heating, increasing drag, structural integrity and aerodynamic stability. The problem of shape optimization is a very old problem and is being targeted by many researchers but an optimum shape design is possibly still not achieved. This study is a step in that direction by using automated computer based optimization by combining Computational Fluid Dynamics (CFD) with optimization codes. The wave drag is the prominent contributor to the drag force at hypersonic reentry velocities. Euler equation coupled with mass and energy conservation equations of fluid dynamics are used as they provide good estimate of wave drag and are computationally efficient compared to Navier-Stokes equations. In this paper, in-house Euler based high resolution WENO (Weighted Essentially Non-Oscillatory) code is used to calculate the wave drag force on 2D reentry module. The code begins with a circular non-dimensionalized geometry having a volume of unity and optimizes the geometry defined by algebraic equations generated by using multiquadric radial basis functions (RBFs) based on the results obtained by CFD. The optimization code uses Lagrangian multipliers for optimization of the reconstructed algebraic equations. The results obtained by the optimizer are further refined in the multi-dimensional parametric domain in the neighborhood of the solution. Finally the optimized solution (shape) is validated by using the commercial CFD software Ansys Fluent 14.