SPACE DEBRIS SYMPOSIUM (A6)
Modelling and Orbit Determination (9)

Author: Dr. Piyush Mehta
University of Strathclyde, United Kingdom, piyush.mehta@strath.ac.uk

Dr. Andrew Walker
Los Alamos National Laboratory, United States, awalker@lanl.gov
Dr. Melrose Brown
Australian Defence Force Academy (ADFA), Australia, Melrose.Brown@adfa.edu.au
Dr. Edmondo Minisci
University of Strathclyde, United Kingdom, edmondo.minisci@strath.ac.uk
Dr. Massimiliano Vasile
University of Strathclyde, United Kingdom, massimiliano.vasile@strath.ac.uk

# MODELING RE-ENTRY AERO-THERMODYNAMICS USING A RESPONSE SURFACE MODEL

**Abstract**

Modeling atmospheric re-entry of debris objects requires knowledge of the aero-thermodynamic properties along the trajectory. Typically, aero-thermodynamic computations in the continuum and free molecular regime are performed using analytical/semi-empirical methods whereas a simple bridging formula, dependent only on the Knudsen number, is used in the transition flow regime. The use of such simple bridging functions result in large errors in computed force coefficients. Also, aerodynamic analytical methods only work well for simple geometries and aerothermodynamic models are only valid for blunt objects with finite curvature radii at the stagnation points . Here we present a novel way of modeling aero-thermodynamics in the transition regime using response surface modeling method. The response surface methodology suffers a much lower impact from the curse of dimensionality and allows for fast evaluation of the model and is therefore chosen over the typical linear interpolation technique. The model is developed for a cube for which analytical aero-thermodynamic methods are not available. High-fidelity simulations for training the model in the transition flow regime are performed using the Direct Simulation Monte Carlo (DSMC) code dsmcFoamStrath, an open source code developed on the OpenFoam framework at the University of Strathclyde. Simulations in the continuum flow regime are performed using the Computational Fluid Dynamics (CFD) code CDF++, developed by Metacomp Technologies. High fidelity simulations at random set of atmospheric properties are used to validate the model.