

INVESTIGATION OF FLOW SEPARATION USING AN ADVANCED COMPUTATIONAL FRAMEWORK

Flow separation is one of the two or three most difficult problems facing the designers of advanced missile/rocket systems. It is responsible for both an increase in drag, a decrease in stability, and a loss of thrust for the vehicle. In turn, these three items are responsible for the majority of the efficiency loss experienced by the system. The problem is most dramatic for the resultant loss in stability since the air frame can be caused to go unstable and, given sufficient time in this state, becomes uncontrollable. In the case of the propulsive system, flow separation can result in such a loss of thrust that the propulsive force is lower than the drag and the system continues to slow down to the point that it cannot meet flight requirements. Hence the need for a predictive dynamic model and design tool for flow separation is driven by some dramatic performance losses and mission failures for missile systems.

A computational investigation of flow separation in advanced supersonic missile/rocket systems including an aft-end jet thruster is being performed. High-fidelity, unsteady, and three-dimensional simulations are being conducted using the proprietary framework *Rocstar*. The computational framework includes a compressible flow solver with advanced LES turbulence models and has flexibility sufficient to investigate a variety of configurations under widely varying conditions. To date, *Rocstar* has been validated for three-dimensional, turbulent, unsteady, multiphase, internal ballistic flows. It has also been validated for solid propellant rocket motors with their ensuing plumes in regimes spanning subsonic to supersonic conditions and has been used to simulate base flows (without thrusters or propulsive plumes) and has compared well with experimental data.

The goal of this work is to develop, validate, and commercialize a numerical tool suitable for design and analysis of flow separation in a coupled jet-interaction and rocket base flow configuration. At the conclusion of the project, we will have completed a study assessing *Rocstar* capabilities for simulating the flow separation problem, and methods for reducing flow separation will be proposed. A student working on this project will learn how to run the large, parallel *Rocstar Simulation Suite* on our dedicated computational cluster, as well as analyze the results of runs to produce data for code validation. Knowledge of basic fluid dynamics would be very helpful to help the student understand the simulations. Knowledge of compressible flow would also be a plus, but is not required.