Computational Fluid Dynamics for the CEV Aerosciences Project

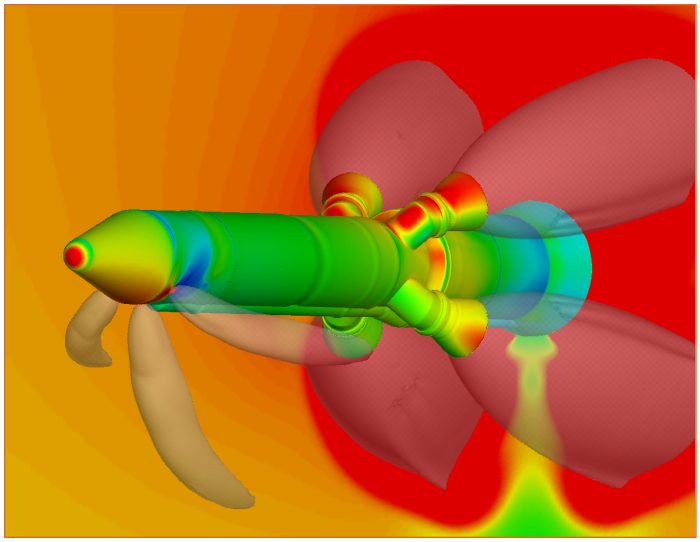
Exploration Systems Mission Directorate

Computational fluid dynamics (CFD) simulations support the design and analysis of NASA's Orion Crew Exploration Vehicle (CEV) and crew module. A range of CFD analyses are conducted to assess key aspects of vehicle performance during launch aborts, atmospheric entry, descent, and landing.

CFD simulations of the Launch Abort Vehicle (LAV) and Orion crew module are used to predict aerodynamic performance and extend wind tunnel test data to flight operating conditions. Analyses of the LAV are also performed to predict abort motor plume expansion and heat transfer at flight operating conditions, and to assess the vehicle's stability and control, including complex interactions between its hardware, attitude control motor, and abort motor plumes. Further simulations model the wake behind the Orion crew module during atmospheric entry and descent to support design and analysis of the parachute deployment system.

These CFD simulations provide the CEV Aerosciences Project with key performance predictions for conditions that are difficult, or impossible, to obtain using ground-based or flight testing. NASA supercomputing resources enable efficient prediction of the aerodynamic performance and flow physics for complex geometries across a complete design space.

Scott Murman, NASA Ames Research Center
scott.m.murman@nasa.gov



Mach contours and isosurfaces highlight the complex flowfield of the Launch Abort Vehicle.
Tom Booth, Ray Gomez, NASA/Johnson