CFD Support for Heavy Lift Launch Vehicles

Exploration Systems Mission Directorate

Computational fluid dynamics (CFD) simulations assist in the design of next-generation heavy lift launch vehicles (HLLVs) and improve our understanding of launch vehicle aerodynamics during ascent. NASA-developed CFD flow solvers are used to predict aerodynamic load distributions on the Ares V HLLV in support of trajectory development and structural analysis. These simulations supplement experimental wind tunnel data and extend predictive capabilities to full-scale flight vehicles.

Extensive analyses of evolving Ares V HLLV designs were performed over several design cycles. Analyses included: validation studies with wind tunnel test data; analysis of aerodynamics throughout the ascent trajectory; investigations of rocket plume effects; shape trade studies to optimize payload shroud design; evaluation of protuberance contributions to aerodynamic loads; solid rocket booster (SRB) separation simulations with plume interactions; and analysis of high-altitude high-Mach conditions that cause plume-induced flow separation and associated risks such as base heating. Prior to this design support work, CFD best practices were developed to assess key analysis criteria such as grid resolutions, turbulence models, discretization schemes, and levels of physical modeling fidelity such as inviscid, viscous, and multi-species models.

Solid rocket booster separation maneuver for Ares V, showing plume isocontours colored by pressure. *Marshall Gusman, NASA/Ames*

NASA's supercomputing resources enable massively parallel computations with high-fidelity models, while powerful postprocessing and visualization tools facilitate interactive examination of complex 3D solutions that would otherwise be too large to inspect. With these capabilities, scientists can quickly compute entire aerodynamic databases and analyze many designs and conditions.

Cetin Kiris, Marshall Gusman, NASA Ames Research Center
cetin.c.kiris@nasa.gov, marshall.r.gusman@nasa.gov