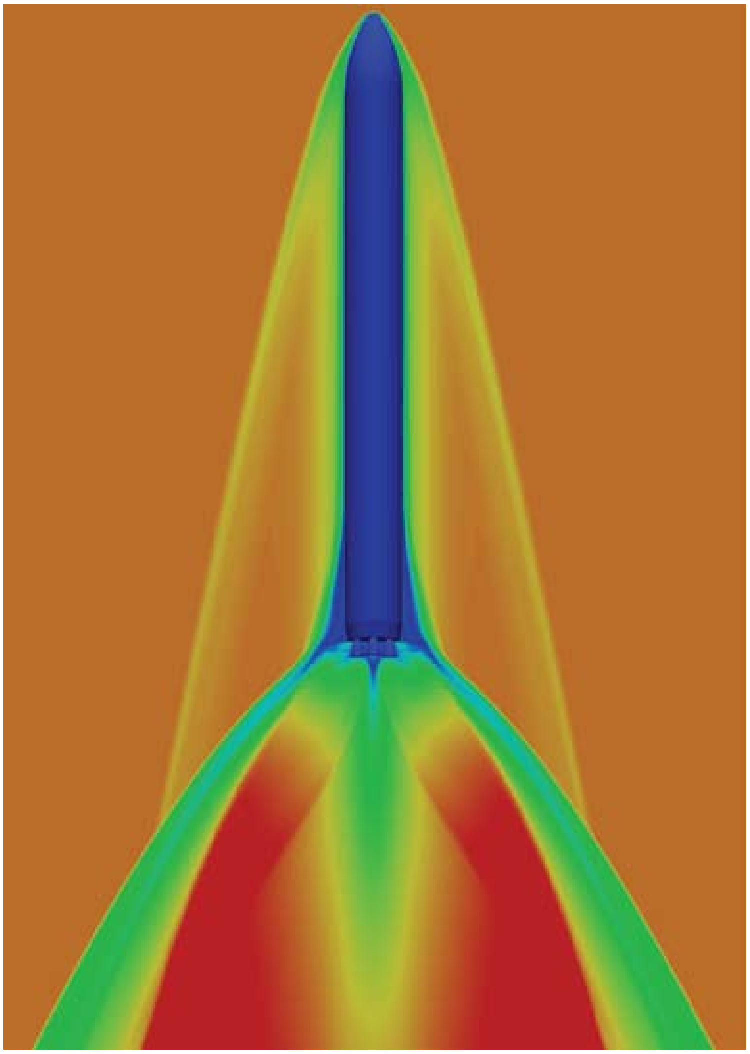
CFD Analysis for the Next-Generation Heavy-Lift Vehicle

Using NASA high-end computing technologies, scientists are producing high-fidelity aerodynamic simulations to assist in the design of the Ares V heavy-lift vehicle that will deliver everything from food and water for astronauts in orbit to heavy scientific payloads bound for the Moon.



Mach contours showing plume-induced flow separation after solid rocket booster separation at high altitude.

Computational fluid dynamics (CFD) analyses of the complete Ares V vehicle have been performed to characterize aerodynamic force and moment coefficients, line loads, and pressure signatures at key instances during the ascent trajectory.

NASA state-of-the-art CFD flow solvers have been used to create a baseline database of integrated force and moment coefficients. Database generation requires hundreds of simulations with significant computational resources, including thousands of processors, large memory and disk space capacities, and concurrent visualization environments.

In the preliminary Ares V design stage, a trade study was conducted to determine the optimal shroud shape to minimize drag and acoustic effects. Vehicle protuberances have been included in the preliminary design cycle analysis. Current CFD best practices for Ares V simulations with and without plume effects have been determined. Analyses that include Solid Rocket Booster and liquid rocket engine plumes have been perfomed to investigate effects such as plume-induced flow separation. This information will contribute to the next design cycle of the launch vehicle and the development of its operational requirements.

These computations are only made possible by the availability of today's supercomputing resources.

Cetin Kiris, NASA Ames Research Center
Cetin.C.Kiris@nasa.gov