High-Fidelity Simulations of Hypersonic Flows

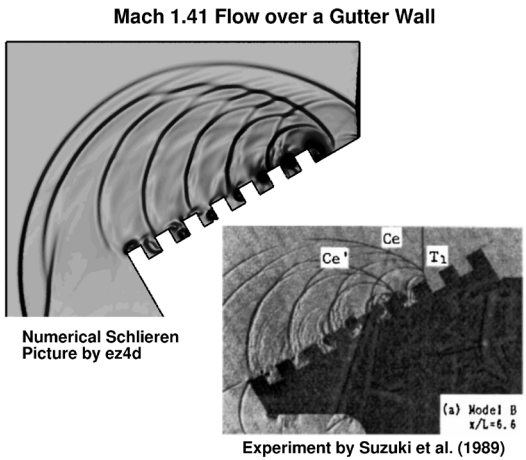
Aeronautics Research Mission Directorate

An advanced computational fluid dynamics (CFD) code called ez4d is being used to simulate complex shock patterns and strong, unsteady waves resulting from surface protuberances and imperfections in the thermal protection systems of NASA's hypersonic vehicles. The ez4d solver is a time-accurate, unstructured mesh, Navier-Stokes code based on the state-of-the-art CESE (conservation element solution element) space-time numerical method. The fundamental physics formulations employed by ez4d enable it to generate high-fidelity numerical solutions for very complex hypersonic flows.

The primary objectives of these simulations are to understand transition flow physics due to roughness elements submerged in a hypersonic boundary layer; predict surface heating caused by transitional and turbulent boundary layers; and improve prediction of wake flows behind large, blunt bodies. These simulations will be used to assist parametric studies for future hypersonic vehicle designs. Another goal of this work is to develop software technologies for future multi-core computing architectures using combined Message Passing Interface (MPI) and multi-thread implementations.

Large-scale computations, using meshes with 10–100 million tetrahedral elements, are being performed on NASA's Pleiades supercomputer, which has enabled reasonable turnaround time for extensive parametric studies that would otherwise be impossible.

Chau-Lyan Chang, NASA Langley Research Center,
Balaji Venkatachari, University of Alabama at Birmingham
chau-lyan.chang@nasa.gov, balaji@uab.edu



Comparison with experimental data for supersonic flow over a gutter wall. *Chau-Lyan Chang, NASA/Langley*