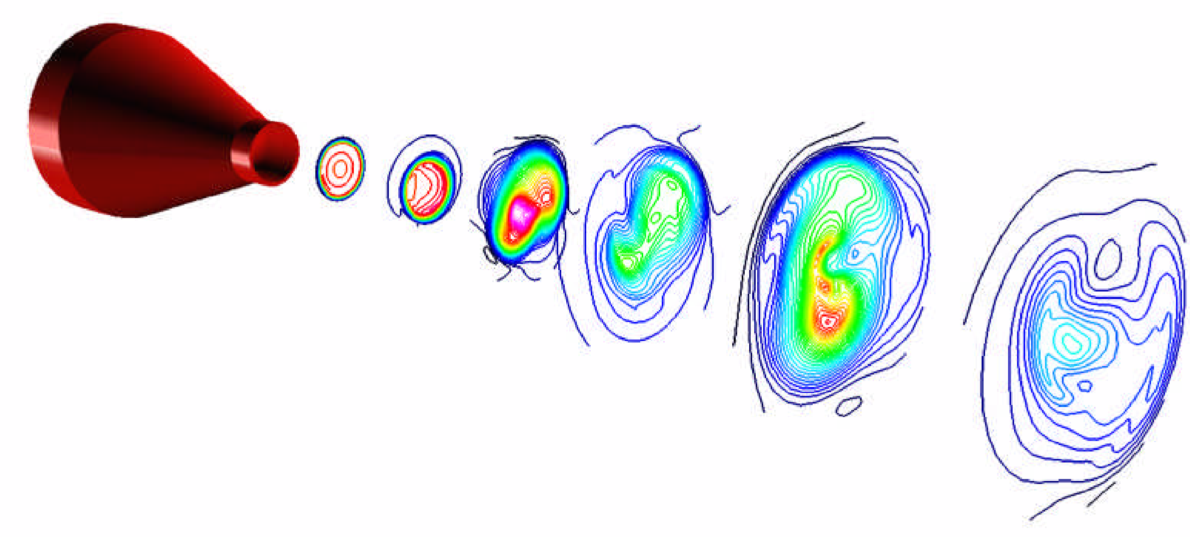
Large-Eddy Simulation Code Developed for Propulsion Applications

A large-eddy simulation (LES) code was developed at the NASA Glenn Research Center to provide more accurate and detailed computational analyses of propulsion flow fields. The accuracy of current computational fluid dynamics (CFD) methods is limited primarily by their inability to properly account for the turbulent motion present in virtually all propulsion flows. Because the efficiency and performance of a propulsion system are highly dependent on the details of this turbulent motion, it is critical for CFD to accurately model it. The LES code promises to give new CFD simulations an advantage over older methods by directly computing the large turbulent eddies, to correctly predict their effect on a propulsion system.

Turbulent motion is a random, unsteady process whose behavior is difficult to predict through computer simulations. Current methods are based on Reynolds-Averaged Navier-Stokes (RANS) analyses that rely on models to represent the effect of turbulence within a flow field. The quality of the results depends on the quality of the model and its applicability to the type of flow field being studied. LES promises to be more accurate because it drastically reduces the amount of modeling necessary. It is the logical step toward improving turbulent flow predictions. In LES, the large-scale dominant turbulent motion is computed directly, leaving only the less significant small turbulent scales to be modeled. As part of the prediction, the LES method generates detailed information on the turbulence itself, providing important information for other applications, such as aeroacoustics.

The LES code developed at Glenn for propulsion flow fields is being used to both analyze propulsion system components and test improved LES algorithms (subgrid-scale models, filters, and numerical schemes). The code solves the compressible Favre-filtered Navier-Stokes equations using an explicit fourth-order accurate numerical scheme, it incorporates a compressible form of Smagorinsky’s model for the subgrid-scale turbulence, and it uses generalized curvilinear coordinates to allow analysis of a wide range of geometries. The code runs in parallel on shared memory multiprocessor computers and is written in Fortran 90 with dynamic memory allocation.

A sample result for a Mach-1.4 round jet is presented in the figure. Instantaneous Mach number contours in several cross-planes downstream of the nozzle exit are shown, illustrating how an LES captures the large unsteady three-dimensional turbulent structures present in the jet.



Cross-plane Mach number contours for a Mach 1.4 jet.

References

1. DeBonis, James R.; and Scott, James N.: Large-Eddy Simulation of a Turbulent Compressible Round Jet. AIAA J., vol. 40, no. 7, 2002, pp. 1346-1354.

2. DeBonis, James R.; and Scott, James N.: Study of the Error and Efficiency of Numerical Schemes for Computational Aeroacoustics. AIAA J., vol. 40, no. 2, 2002, pp. 227-234.

3. DeBonis, James R.: The Numerical Analysis of a Turbulent Compressible Jet. NASA/TM--2001-210716, 2001.

Glenn contact: Dr. James R. DeBonis, 216-433-6581, James.R.DeBonis@nasa.gov
Authors: Dr. James R. DeBonis

Headquarters program office: OAT
Programs/Projects: Vehicle Systems, Propulsion and Power