Advanced turbulent flow modeling for accurate aerodynamic predictions

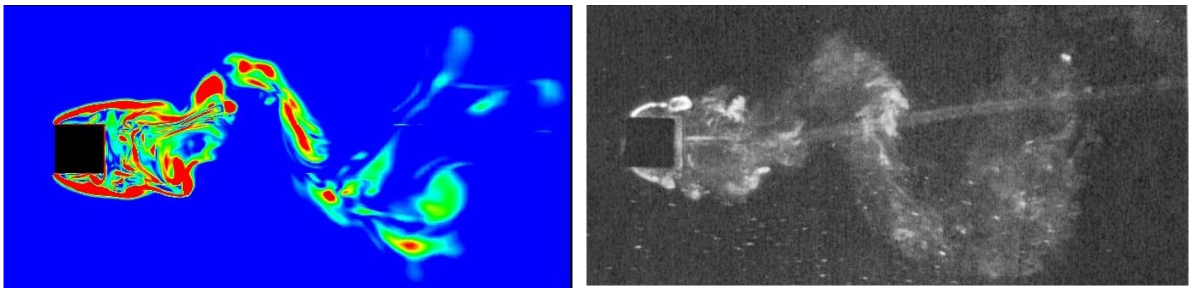


Figure 1: (Left) Instantaneous contours of vorticity magnitude given by a simulation of the turbulent low-speed wake of a square cylinder. (Right) Smoke visualization of the turbulent wake from a water tunnel experiment (Durao et al., Experiments in Fluids, 6(5):298, 1988).

Advances in computer simulations now enable accurate predictions of aerodynamic forces and heating rates

Technical Contact:
Matthew Barone
505-284-8686
mbarone@sandia.gov

Science Matters Contact:
Alan Burns, Ph.D.
505-844-9642
aburns@sandia.gov

Although turbulent fluid motion is encountered in countless engineering devices, its complex dynamical behavior has largely defied attempts at a general theoretical description. To tackle this important problem, aeroscience researchers at Sandia are turning to temporally-resolved computer simulations of compressible turbulent flow as a means of improving engineering predictions of aerodynamic forces and heating rates on flight vehicles. Recent advances in computing power have enabled such simulations to tackle the leap from academic research tool to engineering analysis tool.

Sandia's research focuses on development of accurate numerical simulation techniques for compressible gas flows, where gas velocities may approach or exceed the speed of sound. The larger, more energetic, eddy motions of the gas are directly resolved by the simulation, while the smaller, less energetic, turbulent motions are modeled. Results from a validation study involving the low-speed flow of air past a square cylinder (shown in Figures 1 and 2) illustrate the success of this approach for a bluff-body flow. At flight speeds above the speed of sound, a shock wave precedes the bluff body, presenting a severe challenge to the numerical techniques. Figure 3 (see next page) shows the simulation of an Apollo reentry capsule, demonstrating the ability of the simulation to capture the steady bow shock in addition to the unsteady separated wake surrounding the capsule afterbody.

The development of unsteady turbulence simulation technology will allow analysis of weapon delivery system aerodynamics at an unprecedented level of fidelity. It also opens the door to accurate analysis of many other types of complex, unsteady turbulent flows, from wind turbine blades to tractor-trailer vehicles.

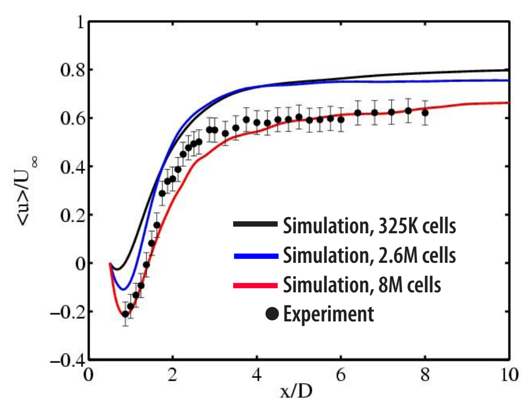


Figure 2. Simulation predictions of the time-averaged streamwise velocity in the square cylinder wake, compared with experimental data.

Sandia National Laboratories

National Nuclear Security Administration

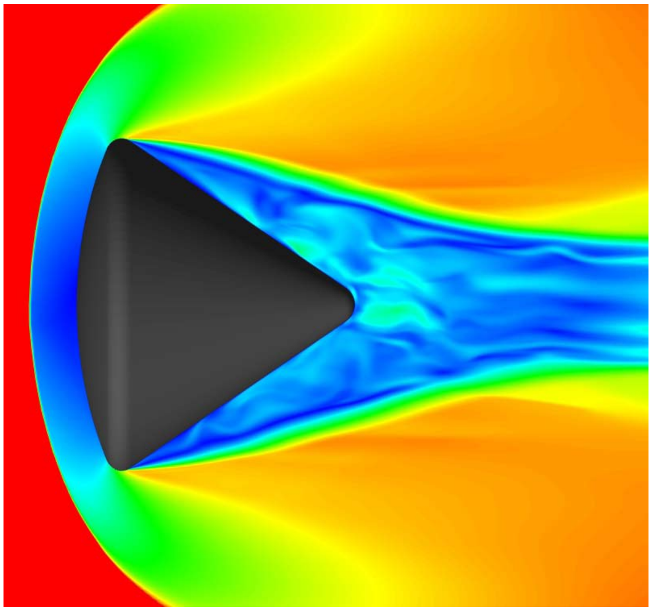
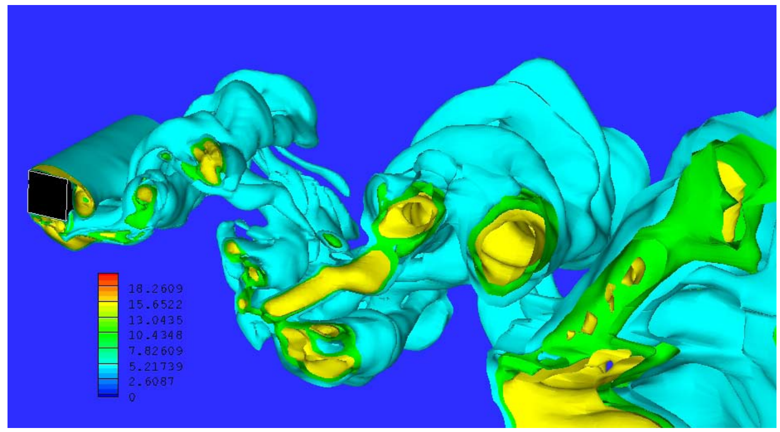


Figure 3. Instantaneous Mach number contours for the flow about an Apollo re-entry capsule, showing the steady bow shock and the turbulent fluctuations in the afterbody wake.



Isosurfaces of the modeled turbulent eddy viscosity in the wake of a square cylinder.

Publications:

M. Barone, "Mesh-independent unsteady turbulent wake simulations using the PANS model," AIAA Paper 2006-3742, 36th AIAA Fluid Dynamics Meeting, San Francisco, CA, June 5-8, 2006.

M. Barone and C. J. Roy, "Evaluation of Detached Eddy Simulation for Turbulent Wake Applications," AIAA J., 44(12):3062-3071, 2006.

Sandia National Laboratories

6/2007

Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000. SAND 2007-3337P

National Nuclear Security Administration