Computational Fluid Dynamics Simulation of Shock-Wave Turbulent Boundary Layer Interactions

Doyle D. Knight

Dept of Mechanical and Aerospace Engineering
Rutgers - The State University of New Jersey
98 Brett Road
Piscataway, NJ 08854-8058 USA
E-mail: knight@soemail.rutgers.edu

Design of high performance supersonic and hypersonic air vehicles requires accurate simulation methods for prediction of aerothermodynamic loads including mean and root-mean-square fluctuating surface pressure, skin friction and heat transfer. Shock wave-turbulent boundary layer interactions can significantly aect aerothermodynamic loads, and therefore accurate methods for their prediction are needed. Recently, an evaluation of Computational Fluid Dynamics (CFD) capability for prediction of shock wave turbulent boundary layer interactions was performed under the auspices of NATO RTO Working Group 10. Five dierent flow configurations were examined by an international collboration of researchers. Three of the configurations were nominally two-dimensional: a compression corner generated by a ramp, an expansion corner followed by a compression corner, and a planar shock wave impinging on a flat plate. The remaining configurations were three dimensional: a single fin attached normal to a flat plate, and two fins attached normal to a flat plate and forming a converging channel. Computational Fluid Dynamics capability for prediction of the detailed flow structure and aerothermodynamic loads was evaluated on the basis of advanced Reynolds-Averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) models through comparison of the predicted results with experiment. The results of this evaluation will be presented, and directions for future research will be discussed.