PRELIMINARY LES OVER A HYPERSONIC ELLIPTICAL CROSS-SECTION CONE

M.P. MARTIN
Mechanical and Aerospace Engineering, Princeton University, Princeton, NJ 08544

M. WRIGHT
Eloret Corporation, NASA Ames Research Center, Moffett Field, CA 94035

G.V. CANDLER
Aerospace Engineering and Mechanics, University of Minnesota, Minneapolis, MN 55455

U. PIOMELLI
Mechanical Engineering, University of Maryland, College Park, MD 20742

G. WEIRS
ASCI Flash Center, University of Chicago, Chicago, IL 60637

AND

H. JOHNSON
Aerospace Engineering and Mechanics, University of Minnesota, Minneapolis, MN 55455

Abstract. The characteristics of the hypersonic flow around an elliptical-cross section cone and the computational code to perform large-eddy simulations of this flow are described. Preliminary results of the Lagrangian implementation used to compute the subgrid-scale terms are presented.

1. Introduction

Many aspects of transitional and turbulent flows are not fully understood. This is especially true in the hypersonic regime, where examples of unresolved issues include the effect of freestream disturbances and threedimensionality. In the absence of detailed experimental or computational databases to better understand these physical phenomena, we are left with excessive design conservatism and unrefined conceptual designs.

When investigating these phenomena via CFD, direct numerical simulations (DNS) are not affordable. Turbulence models provide a wide range of accuracy in predicting turbulent flows of engineering interest. Depending on the level of detail required, one may choose the Reynolds-averaged Navier-Stokes (RANS) models or the state of the art subgrid scale (SGS) models in a large-eddy simulation (LES) to obtain a more refined prediction. The study of fundamental physical phenomena must be done using the finest possible prediction, namely LES. However, one must keep in mind that a key feature of the prediction is the validation with experiments.

Using the most recent laser and camera technologies, Huntley[1] and Huntley *et al.*[2] present the first detailed flow visualization of natural transition on an elliptical cross-section cone at Mach 8. Mean flow features and details about the unstable modes in the boundary layer for the same configuration are given by Kimmel *et al.*[3,4] and Poggie *et al.*,[5] respectively. Because this flow is being extensively documented experimentally and because the geometric configuration resembles that of the forebody of a hypersonic vehicle, this database is ideal to test the state of the art SGS models and LES methodology for hypersonic flows.

The present work is an ongoing effort to provide detailed flow simulations of unsteady, hypersonic, transitional or turbulent flows. In Martin *et al.* (2000a) we develop and test SGS models for compressible LES using the apriori test in compressible, isotropic turbulent flow. In Martin *et al.* (2001) we validate the LES methodology against the DNS of supersonic boundary layers, and in Martin *et al.* (2000b) we extend the LES code to generalized curvilinear coordinates and validate the implementation in supersonic turbulent boundary layer flow. In this paper, we present preliminary LES results of the flow around a section of an elliptical cross-section cone away from the tip of the cone. The flow conditions, simulation procedure, preliminary flow assessments, and future work are given in the following sections.

2. Geometry and flow conditions

Following the experimental configuration of Huntley *et al.*,[2] the lifting body geometry is a sharp-nosed, elliptical cross-section cone. The nominal radius of the experimental model is less than 200 μm. In our simulations, we use an ellipsoidal nose with an 80 μm diameter of nose ellipsoid measured on the major axis. The afterbody is an elliptical cross-section cone of 4:1 aspect ratio, 17.5 degrees half-angle in the major axis, and 0.242 m in length, resulting in base dimensions of 0.152 m across the major axis and 0.038 m across the minor axis.

The freestream flow conditions are $Re_\infty = 14 \times 10^6\ /m$, $M_\infty = 8$, $\rho_\infty = 0.5$, and $T_\infty = 58$ K. For these conditions, the boundary layer is fully turbulent at position x=17.5 cm from the nose.[1] The wall-temperature condition is prescribed to 450 K, which is nearly adiabatic. As in the experiments, air is the working fluid.

The Mach 8 flow around the 4:1 elliptical cross-section cone at zero angle of attack is highly compressed behind the shock. The difference in shock strength between the major and minor axis causes a higher compression at the leading edge producing a cross flow from the leading edge to the centerline of the cone. This is illustrated in Fig. 1a. At the centerline the cross-flow velocities are zero and mass conservation induces a bulge, see Fig. 1b, where the boundary layer is twice as thick as the boundary layer in the off-center region.[1] Experiments show that transition occurs first on the centerline.[1]

3. Grid resolution

We estimate the resolution requirements by considering the dimensions of the turbulence structure and the data provided by the experiments. From the experimental data[1], the estimated wall unit is 7.5×10^{-5} m and the boundary layer along the centerline is laminar at $x = 9$ cm, late-transitional at 11.4 cm, and fully turbulent at 17.5 cm from the nose. See Figure 2. The turbulent boundary layer at the center line is 5 to 6 mm thick. Note that the shock-standoff distance is $\theta_s = 1.2\theta_c$, where θ_s and θ_c are the shock and cone angles, respectively. The turbulence structures on a flat plate boundary layer extend about 100 wall units in the spanwise direction and a few boundary layer thicknesses in the streamwise direction.

In the spanwise direction we use a uniform grid spacing of 22 wall units. This estimate is based on the grid resolution used in previous LES of a supersonic boundary layer.[7] For the conditions chosen, the boundary layer at the leadingedge is laminar.[1] Since the flow is supersonic and the cone is at zero angle of attack, the flow around the top of the cone is not affected by the flow on the bottom. Thus only the top of the cone is simulated. In the streamwise direction, the flow is laminar at $x = 9$ cm from the nose. Thus, the resolution requirements up to 9 cm from the nose are given by the grid convergence studies of the laminar flow at the nose. From $x = 9$ cm to the base of the cone we require that the maximum grid spacing on the surface is 33 wall units. In the wall-normal direction the grid is exponentially stretched, we require 0.3 and δ^+ wall units for the minimum and maximum grid spacings within the boundary layer,[7] where $\delta^+ = \delta/z_\tau$ is about 70 for the turbulence case. The computational domain is large enough to include the standoff shock wave.

4. Simulation procedure

For the LES, we use a third-order accurate WENO[9] to compute the convective fluxes. This scheme has low dissipation properties, and was designed to perform DNS and LES of compressible flows. The time advancement technique is based on the DPLU relaxation method of Candler *et. al*[10] and was extended to second-order accuracy by Olejniczak and Candler.[11] The viscous fluxes are evaluated using fourth-order central differences. Finally, the transformation metrics are evaluated using fourth-order central differences so that the inaccuracy of the numerical evaluation of the metrics coefficients is less than the inaccuracy of the convected fluxes.

The initial condition will be a superposition of laminar flow and a prescribed freestream energy disturbance spectrum. To generate the laminar solution we use a finite volume code,[12] where we only compute 90 degrees of the cone geometry and use bilateral symmetry to reflect the resulting solution across the centerline and generate the full 180 degrees. We then interpolate the laminar solution from the finite-volume cell centers to the finite-difference grid points using tri-linear interpolation.

The required SGS terms and the model representations are given in Martin *et al.* (2000a, 2000b). To evaluate the model coefficients we use the Lagrangian-averaging operation, where the averaging is performed along a fluid particle pathline. A full description of the Lagrangian-average procedure can be found in Meneveau *et al.* (1996).

5. Preliminary flow assessments and future work

In this section we present a brief progress report in performing the LES. A portion of the cone has been initialized using the interpolated laminar solution to test the Lagrangian implementation of the SGS models. To minimize the complexity of this test, the disturbances introduced by the tri-linear interpolation are used as initial disturbances. Figures 3 through 5 show contours of the SGS terms at the exit plane of the cone on spanwise wall-normal planes. These figures include a quarter of the computational domain (centered about the centerline) in the spanwise direction, and about ten boundary layer thickness in the wall normal direction, the boundary layer thickness along the centerline is about 2 mm, three times smaller than the turbulent boundary layer. Figures 3 through 5 show that the SGS terms are dominant in the bulge region. This would not be the case if the model coefficients were calculated using the ensemble average procedure.

The magnitude of the initial disturbances is very small and do not grow significantly in time. Future work includes imposing a prescribed disturbance energy spectrum and running the LES to gather sufficient statistical data to assess the effect of freestream disturbance and study the transition and turbulence phenomena.

Computer time was provided by NASA Ames Research Center. This work was supported in part by the Air Force Office of Scientific Research under grant AF/F49620-01-1-0060.

6. References

[1] Huntley, M., "Transition on elliptic cones at Mach 8," PhD thesis, Princeton University, November 2000.
[2] Huntley, M., Wu, P., Miles, R.B., Smits, A.J., "MHZ Rate Imaging of Boundary Layer Transition on Elliptic Cones at Mach 8," *AIAA Paper No. 2000-0379*, January 2000.
[3] Kimmel, R.L., and Poggie, J., "Laminar-Turbulent Transition in a Mach 8 Elliptic Cone Flow," *AIAA Journal,* **37**, 1080, 1999.
[4] Kimmel, R.L., Klein, M.A., and Schwoerke, S.N., "Three-Dimensional Hypersonic Laminar Boundary-Layer Computations for Transition Experiment Design," *Journal of Spacecraft and Rockets,* **34**, 409, 1997.
[5] Poggie, J., Kimmel, R.L., and Schwoerke, S.N., "Traveling Instability Waves in a Mach 8 Flow over an Elliptic Cone," *AIAA Journal,* **2**, 2000.
[6] Martín, M.P., U. Piomelli and G.V. Candler, "Subgrid-Scale Models for Compressible Large-Eddy Simulations," *Theoretical and Computational Fluid Dynamics*, **13**, 361, 2000a.
[7] Martín, M.P., U. Piomelli and G.V. Candler, "Direct and large-eddy simulations of turbulent supersonic boundary layers," *Submitted to Journal of Fluid Mechanics,* 2001.
[8] Martín, M.P., G. Weirs, G.V. Candler, U. Piomelli, H. Johnson, I. Nompelis, "Toward the large-eddy simulation over a hypersonic elliptical cross-section cone," *AIAA Paper No. 00-2311*, June 2000b.
[9] Weirs, V.G., and G.V. Candler, "Optimization of weighted ENO schemes for DNS of compressible turbulence," *AIAA Paper No. 97-1940*, June 1997.
[10] Candler, G.V., Wright, W.J. & McDonald, J.D. *AIAA Journal,* 32, 2380-2386, 1994.
[11] Olejniczak, D.J. , and G.V. Candler, "A Data-Parallel LU relaxation method for DNS of compressible flows," *1st International Conference in DNS and LES*, Louisiana, August 1997.
[12] Wright, M.J., G.V. Candler, and D. Bose, "Data-Parallel Line Relaxation Method for the Navier-Stokes Equations," *AIAA Journal*, **36**, 9, 1998.
[13] Meneveau, C., T.S. Lund and H.Cabot, "A Lagrangian Dynamic Subgrid-scale Model of Turbulence," *Journal of Fluid Mechanics,* **319,** 353-385, 1996.

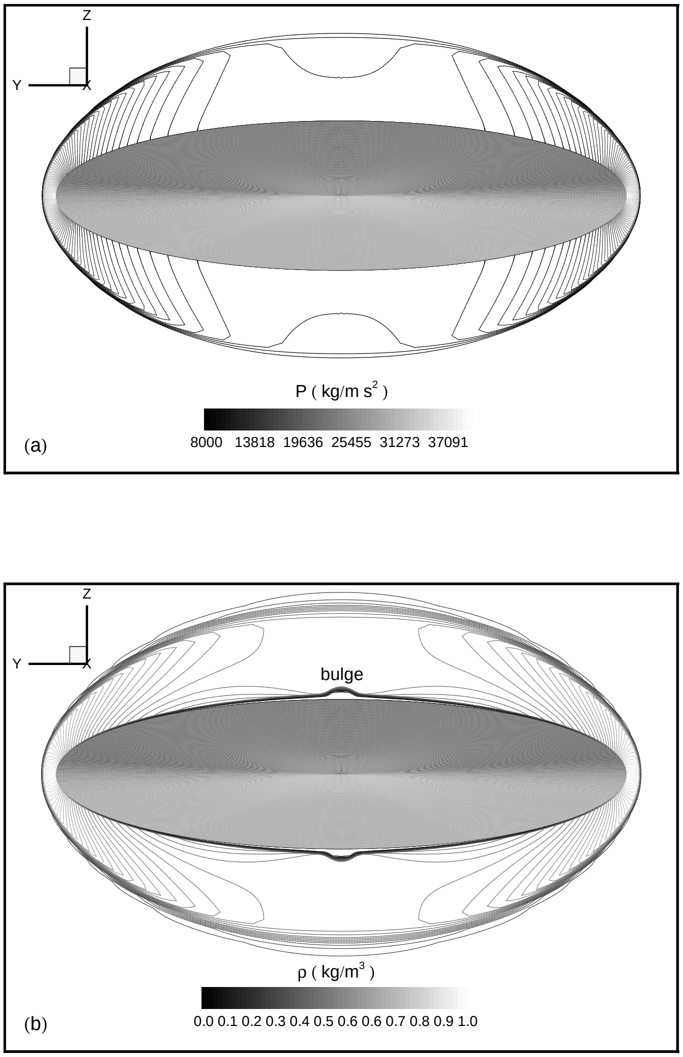

Figure 1. (a) Pressure and (b) density contours in the exit plane of the cone geometry for the laminar solution.

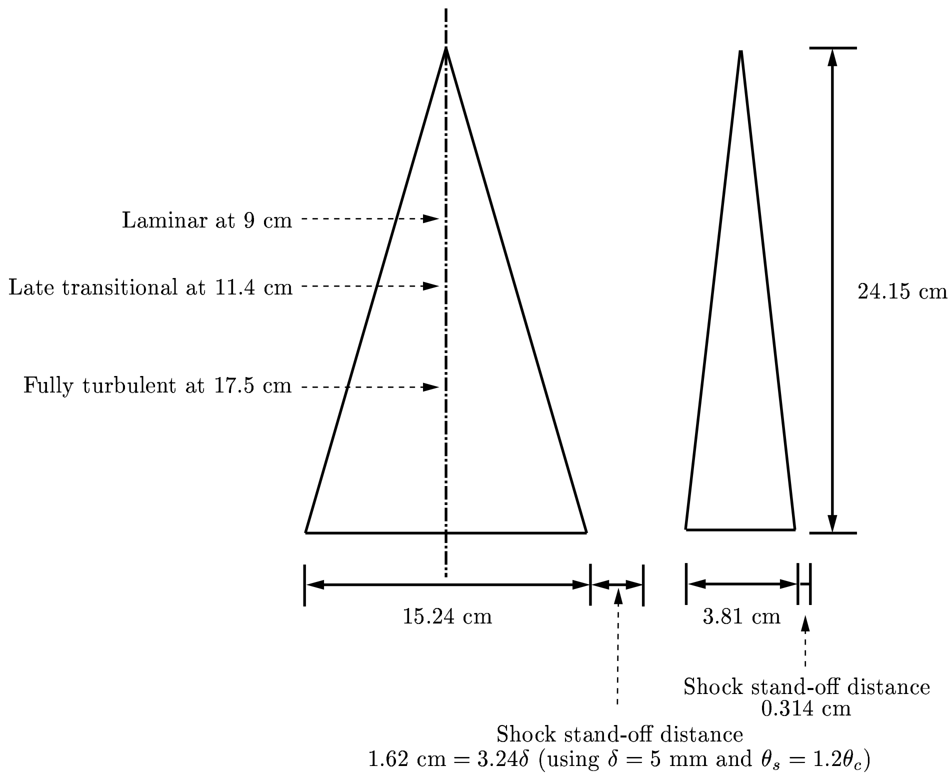

Figure 2. Model dimensions and key flow features required to determine the grid resolution and computational domain size.

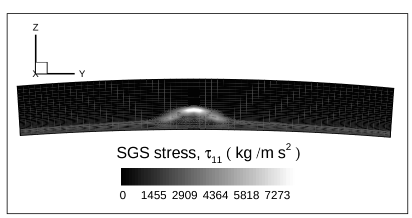

Figure 3. SGS stress contours on the exit plane, the flow is into the page.

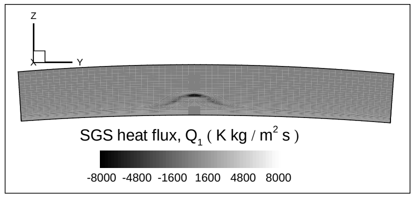

Figure 4. SGS heat flux contours on the exit plane, the flow is into the page.

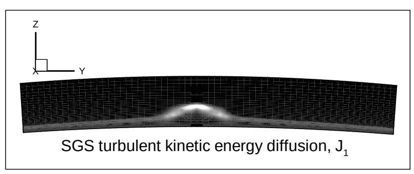

Figure 5. Contours of SGS turbulent kinetic energy on the exit plane, the flow is into the page.