References

[1]Eidelman, S., "The Problem of Gradual Opening in Wave Rotor Passage," *Journal of Propulsion and Power*, Vol. 1, Jan.-Feb. 1985, pp. 23-28.

[2]Klapproth, J.F., "Supercharged Turbowave Engines," Flight Propulsion Division, GE, Cincinnati, Tech. Rept. R62FPD171, May 1962.

[3]Pearson, R.D., "Pressure Exchanges and Pressure Exchange Engines," *Thermodynamics and Gas Dynamics of E.C. Engines*, by R.S. Benson (to be published), Chap. 16.

Real Gas Effects on the Numerical Simulation of a Hypersonic Inlet

Wing-Fai Ng*
Virginia Polytechnic Institute and State University
Blacksburg, Virginia
and
Thomas J. Benson† and William G. Kunik†
NASA Lewis Research Center, Cleveland, Ohio

Introduction

THE aerodynamic performance of the air-induction system is an important factor in establishing the viability of a hypersonic vehicle design. Since the inlet performance is primarily a function of the internal contour geometry, the development and assessment of analytical techniques for the design of internal contours are items of primary concern. The inlet design technology must have a detailed and accurate flowfield calculation procedure that includes the effect of the boundary layer. Although a full Navier-Stokes procedure provides the necessary generality to predict the flow in an inlet, the required computer time and storage indicate that such a procedure should be used only if no suitable alternative exists.[1] An optimum analysis possesses the general viscous nature of the Navier-Stokes equations, but takes advantage of realistic physical approximations to limit the computer running time and storage requirements associated with the solution of the complete Navier-Stokes equations. One approach is to use a spatial-marching procedure, which reduces the complete Navier-Stokes equations to a form that can be treated as an initial boundary value problem and solved by forward marching in space. The assumption made is that a primary flow direction exists and that diffusion in this direction can be neglected. In this manner, a set of steady-state equations is produced for entirely supersonic flows that can be solved by an efficient spatial-marching procedure. In any embedded subsonic regions, such as at no-slip walls, further approximations are required to allow solution by spatial marching.

Description of Problem

Hypersonic Inlet

The problem considered is the numerical simulation of the flowfield in a two-dimensional, high-speed inlet. The geometry of the inlet is shown in Fig. 1. The model exhibits the significant characteristics of hypersonic inlets, for which extensive experimental data have been documented.[2] The model, denoted as P8, represents inlet configurations typical of cruise condition of a hypersonic airbreathing vehicle. It was designed to provide an internal compression ratio of 8. The forebody wedge is a nominal 6.5 deg, intended to match a design Mach number of 6 at the inlet entrance under the test conditions of a freestream Mach number of 7.4, allowing for boundary-layer displacement effects. The wedge was cooled, providing a relatively uniform surface temperature of 0.375 $T_{t\infty}$, where the freestream total temperature $T_{t\infty}$ is 811 K. The freestream Reynolds number is 8.86×10^6 m^{-1}. The boundary-layer transition point was found experimentally to be at approximately 40% of the distance between the wedge leading edge and the inlet entrance.

The cowl was designed with a leading-edge diameter of 0.114 cm and the cowl was kept at constant temperature of 0.375 $T_{t\infty}$. For the P8 inlet, the cowl boundary-layer transitions are located approximately halfway between the cowl leading edge and the throat station.

The inlet configuration, with realistic geometry, important viscous-inviscid interactions, and extensive experimental data, provides an excellent opportunity to verify the efficacy of the numerical method and to understand the complex phenomena of high-speed inlets.

Real-Gas Effects

At hypersonic speed, temperature changes across the shock wave can be so high that the imperfect-gas effects become important. The gas can be considered to be thermally perfect (i.e., $Pv = RT$), but calorically imperfect (i.e., specific heats are not constant, but a function of temperature). The effect of variable specific heats can be illustrated in Fig. 2, in which the total pressure ratio across a normal shock wave is plotted against the Mach number for three different values of the ratio of specific heats, corresponding to upstream static temperature of 500, 1000, and 1500 R. It can be seen that if the normal shock occurs at a Mach number of 5, the difference in total pressure loss across the shock is more than 20% for $\gamma = 1.4$ (500 R) and

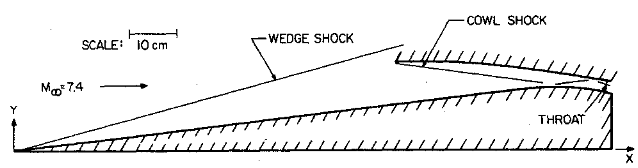

Fig. 1 Hypersonic inlet geometry.

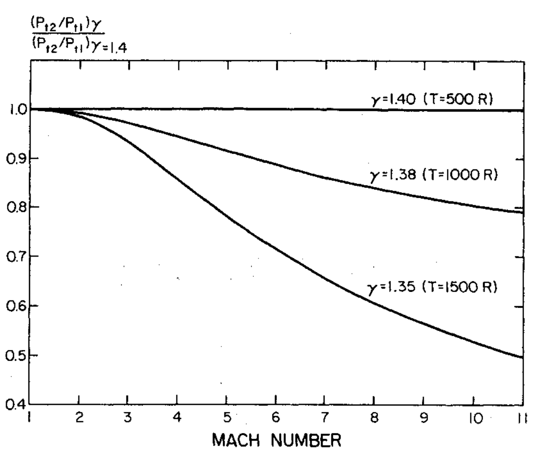

Fig. 2 Effect of caloric imperfections on the total pressure across a normal shock wave.

Received Jan. 9, 1986; revision received March 15, 1986.

*Assistant Professor, Department of Mechanical Engineering. Member AIAA.

†Aerospace Engineer, Computational Applications Branch, Internal Fluid Mechanics Division.

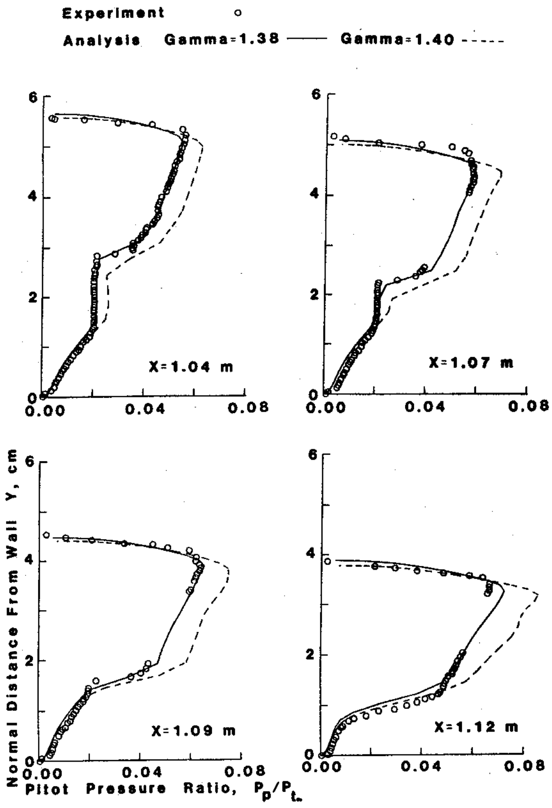

Fig. 3 Comparison of pitot pressure.

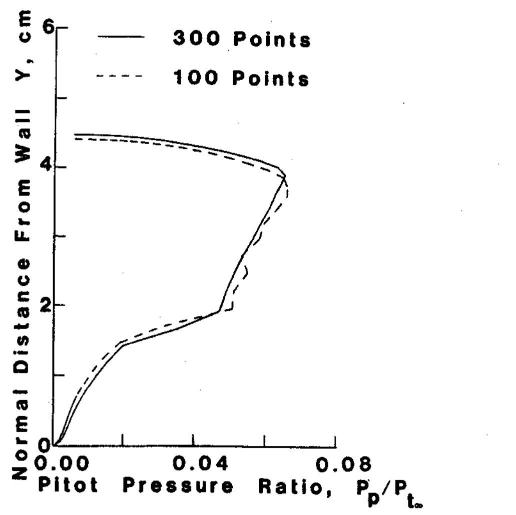

Fig. 4 Effect of radial grid points on performance prediction.

$\gamma = 1.35$ (1500 R). Figure 2 demonstrates the significance of the real-gas effects in a hypersonic flowfield.

Performance Calculation

Investigations were performed to assess the real-gas effects on a realistic hypersonic inlet geometry. The parabolized Navier-Stokes solver was used to simulate the flowfield for the P8 inlet. The code used is a viscous marching analysis (PEPSIS). The solution methodology, or calculational protocol, is documented in detail in Ref. 3. Sensitivity studies in areas such as the initial conditions, wall functions, and turbulence model on the calculated results will be reported in a subsequent paper. Two cases were investigated in the current effort: constant ratio of specific heats $\gamma = 1.40$ and 1.38, corresponding to a freestream static temperature of 500 and 1000 R, respectively. It is to be emphasized that the calulation was done by assuming a constant ratio of the specific heats in the entire flowfield. The objective is to perform a sensitivity study that will bracket the real-gas effects on calculations of the hypersonic inlet performance. A further effort is underway to make the code capable of accounting for changes in the specific heats in the flowfield caused by the temperature change across the shock wave.

The results are summarized in Fig. 3, in which the comparison between experiment and analysis is presented in terms of the pitot pressure ratio vs the normal distance from the centerbody. Comparisons are made at four different axial locations from $x = 1.07$ to 1.12 m (see Fig. 1). The significant effect due to the change in γ can be seen easily. Although the change in the ratio of specific heats is less than 1.5% (which is realistic considering the change in the temperature across the first shock from the cowl lid), the resultant change in the calculated pitot pressure ratio is more than 15%.

The close agreement between experiment and analysis (for $\gamma = 1.38$) demonstrated the capability of a simplified Navier-Stokes solver to simulate the flowfield in a realistic hypersonic inlet. The considerable savings in computer time and storage requirements compared to a full Navier-Stokes solver can make this a powerful tool for understanding the complex fluid physics in hypersonic inlets. With further development and optimization, the code can become a useful design tool.

A parametric study on the effect of the number of transverse direction grid points on the performance prediction was performed. Cases were run at 100, 200, and 300 radial grid points. Not much difference is observed between 200 and 300 grid points. However, significant suppression of the postshock oscillation between 300 (or 200) and 100 grid points can be seen in Fig. 4. The calculations had significantly more grid points within the boundary layers, leaving marginally enough points in the freestream for the case with 100 grid points. The result presented in Fig. 3 was computed with 300 points.

The present study was performed on a Cray-1S computer using 0.4×10^6 bytes of memory and 15 min of CPU time for a 200×3000 mesh. This is relatively inexpensive to run, especially when compared to a full Navier-Stokes solver with similar grid points.

Conclusions

The feasibility of using a simplified Navier-Stokes solver (parabolized) to accurately simulate the flowfield in a hypersonic inlet has been demonstrated. Good agreement is obtained between the numerical analysis and experiment for a Mach 7.4 inlet under cruise condition with an internal compression ratio of 8. The studies also reveal the significance of real-gas effects on the performance calculation of hypersonic inlet. A small change in the ratio of specific heats can result in a significant change in the calculated pitot pressure ratio. This suggests that future efforts in the numerical simulation of hypersonic flowfield must account for the real-gas effects in the computation code.

Acknowledgment

The authors are indebted to Dr. L. A. Povinelli for his support in this research.

References

[1]Knight, D. D., "Numerical Simulation of Realistic High-Speed Inlets Using the Navier-Stokes Equations," *AIAA Journal*, Vol. 15, Nov. 1977, pp. 1583-1589.

[2]Seebaugh, W. R., "Hypersonic Flows in Large-Scale Inlet Models," *Journal of Aircraft*, Vol. 10, Jan. 1973, pp. 38-44.

[3]Buggeln, R. C., Kim, Y. N., and McDonald, H., "Computation of Multi-dimensional Viscous Supersonic Flow," NASA CR, in preparation.