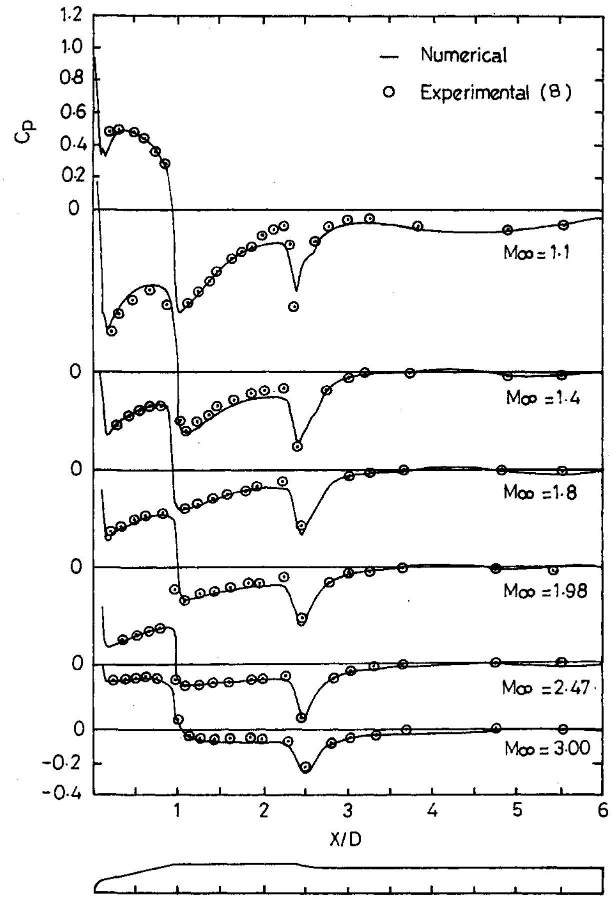

Fig. 6 Pressure distribution along heat shield; $M \leq 3.0$.

forebody and the boat-tail region reduces with increasing supersonic Mach number.

Conclusions

A numerical experiment of axisymmetric turbulent viscous flow over a bulbous heat shield is performed by employing a three-stage Runge–Kutta time-stepping scheme. Turbulence closure is achieved using the Baldwin–Lomax turbulence model. The flowfield visualization of the terminal shock and the separated region helps in a systematic understanding of flow structure under various freestream Mach numbers. The following observations are made based on the described numerical simulations.

1) The terminal shock moves downstream with increasing freestream Mach number. The strength of the terminal shock initially increases with Mach number and later decreases. The location of the terminal shock is found as a nonlinear function of the freestream Mach number.

2) The separation zone in the boat-tail region is found as a function of the freestream Mach number. The flowfield features of the separated region are found to be different in the transonic and supersonic Mach number ranges.

References

[1]Ahmed, S., and Selvarajan, S., "Investigation of Flow on a Hammerhead Nose Configuration at Transonic Speeds," AIAA Paper 91-1711, June 1991.

[2]Ramaswamy, M. A., and Rajendra, G., "Experimental Investigation of Transonic Flow Past a Blunt Cone–Cylinder," *Journal of Spacecraft and Rockets*, Vol. 15, No. 2, 1978, pp. 120–123.

[3]Purohit, S. C., "A Navier–Stokes Solution for a Bulbous Payload Shroud," *Journal of Spacecraft and Rockets*, Vol. 23, No. 6, 1986, pp. 590–596.

[4]Deese, J. E., Gielda, T. J., Agarwal, R. K., and Pavish, D. J., "Prediction of Flow Separation on Launch Vehicle Configuration," AIAA Paper 91-1727, June 1991.

[5]Baldwin, B. S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flow," AIAA Paper 78-257, Jan. 1978.

[6]Peyret, R., and Vivind, H., *Computational Methods for Fluid Flow*, Springer–Verlag, Berlin, 1993, pp. 109–111.

[7]Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solution of Euler Equations by Finite Volume Methods Using Runge–Kutta Time Stepping Schemes," AIAA Paper 81-1259, 1981.

[8]Mehta, R. C., "Transonic Flow Simulation for a Bulbous Heat Shield," *Journal of Spacecraft and Rockets*, Vol. 34, No. 4, 1997, pp. 561–564.

[9]Prasad, J. K., Varambally, B. S., and Pillai, N. M., "Aerodynamic Load on Heat Shield in Mach Number Range of 0.8 to 1.1," Vikram Sarabhai Space Center, AFDG:01:84, Trivandrum, India, June 1984.

[10]Ahmed, S., "Flow Visualization Studies at Transonic Speeds on Heat Shield Configurations," National Aeronautical Labs., NAL-TWT-1-36, Bangalore, India, July 1984.

J. R. Maus
Associate Editor

Euler Solution of Axisymmetric Jets in Supersonic External Flow

Erdal Yılmaz* and Mehmet Şerif Kavsaoğlu†
Middle East Technical University,
06531 Ankara, Turkey

Nomenclature

a = speed of sound, m/s
D = nozzle external diameter, m
$\boldsymbol{F}$ = convective flux vector in x direction, nondimensional
$\boldsymbol{G}$ = convective flux vector in y direction, nondimensional
$\boldsymbol{H}^*$ = axisymmetric source vector, nondimensional
K = node number, nondimensional
M = Mach number, nondimensional
P = pressure, nondimensionalized with P_∞
R = residual, nondimensional/radius, m
r = radial distance, m
S = control volume surface, nondimensionalized with D^2
t = time, nondimensionalized with a_∞ and D
$\boldsymbol{U}$ = solution vector, nondimensional
$\boldsymbol{V}$ = velocity vector, nondimensionalized with a_∞
x, y = two-dimensional or axially symmetric coordinates, nondimensionalized with D
γ = ratio of specific heats, nondimensional
Ω = control volume, nondimensionalized with D^3

Subscripts

j = jet
∞ = infinity

Introduction

An exhaust plume expansion into a supersonic external freestream is a complex phenomenon. The plume boundary behaves as a curved rigid body, and it causes a curved shock wave. Very large gradients exist around these regions.[1] Some experimental studies have been performed to obtain plume behavior at different jet exit pressure ratios.[2–4] There are several methods for predicting exhaust plume boundaries.[1,2,5,6]

In this study, axisymmetric Euler equations were solved by using a finite volume technique on unstructured triangular grids. Jet plume boundaries and other flowfield details were obtained for supersonic axisymmetric jets in a parallel supersonic external flow. The jet-to-freestream pressure ratio varied between 1 and 1500. For very high pressure ratios, computations are started with a low pressure ratio,

Received April 10, 1997; revision received Aug. 25, 1997; accepted for publication Oct. 10, 1997. Copyright © 1997 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

*Ph.D. Student, Aeronautical Engineering Department.

†Associate Professor, Aeronautical Engineering Department. Senior Member AIAA.

and it is increased step by step throughout the iterations to overcome numerical instabilities.

Governing Flow Equations

The vector form of two-dimensional or axisymmetric Euler equations is given as

$$\frac{\partial \boldsymbol{U}}{\partial t}+\frac{\partial \boldsymbol{F}}{\partial x}+\frac{\partial \boldsymbol{G}}{\partial y}+\alpha \boldsymbol{H}^{*}=0 \tag{1}$$

where α is 0 for two-dimensional problems and 1 for axisymmetric problems.

Applying Gauss theorem in the plane, Eq. (1) can be integrated over a control volume Ω as follows:

$$\frac{\partial}{\partial t}\int_{\Omega} \boldsymbol{U}\,\mathrm{d}\Omega+\oint_{S}(\boldsymbol{F}\,\mathrm{d}y-\boldsymbol{G}\,\mathrm{d}x)+\alpha\int_{\Omega}\boldsymbol{H}^{*}\,\mathrm{d}\Omega=0 \tag{2}$$

Numerical Solution Technique

Equation (2) is discretized in space by using a finite volume formulation.[7] Artificial dissipation terms[8] are used to damp oscillations. When Eq. (2) is applied to finite control volumes, surrounding the nodes of the computational grid, a system of coupled ordinary differential equations is obtained in the following form:

$$\frac{\mathrm{d}\boldsymbol{U}_K}{\mathrm{d}t}+R(\boldsymbol{U}_K)=0 \tag{3}$$

The integration in time to a steady-state solution is performed by a multistage explicit scheme.[7,9] Local time stepping, residual averaging, and adaptive remeshing are used for convergence acceleration. The initial and adapted meshes were generated by the advancing front technique. $|\boldsymbol{V}\times\nabla M|$ was selected as the sensor to obtain local grid spacings of the adapted grid.

Computational Details

Computations were performed on the IBM-SP2 machine installed at the Middle East Technical University. For 1000 iterations and for a grid size of 3217 points and 9395 triangular elements, 1.1 megabytes of memory and 700 s of CPU time were required. This CPU time includes writing the solution file after each 100 iterations.

Results

The results are presented for a case with $M_j=3$, $M_\infty=4$, and $P_j/P_\infty=1500$. The ratio of specific heats γ was 1.4 for both jet and external flow. The adapted grid is shown in Fig. 1. The grid was refined around the curved oblique shock and the jet plume boundary and rarefied at the regions of low gradients. The streamlines are presented in Fig. 2; the jet plume boundary prediction is compared with the experimental results in Fig. 3 (Ref. 2); and the predicted plume shapes for various pressure ratios are presented in Fig. 4.

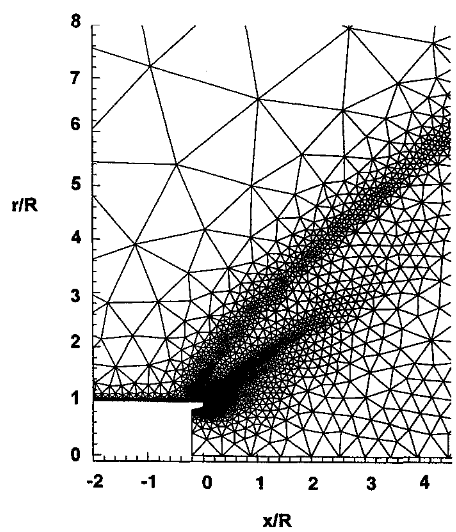

Fig. 1 Adapted mesh: $M_\infty=4$, $M_j=3$, and $P_j/P_\infty=1500$.

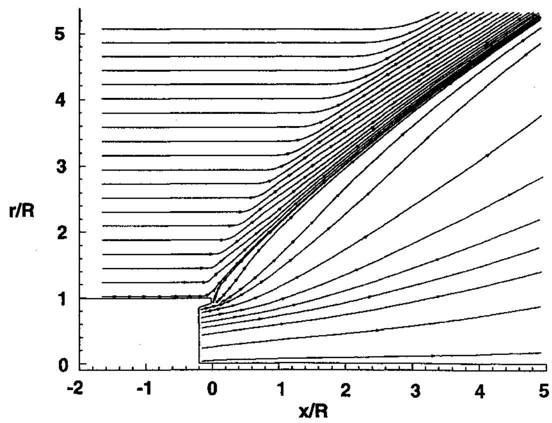

Fig. 2 Streamlines: $M_\infty=4$, $M_j=3$, and $P_j/P_\infty=1500$.

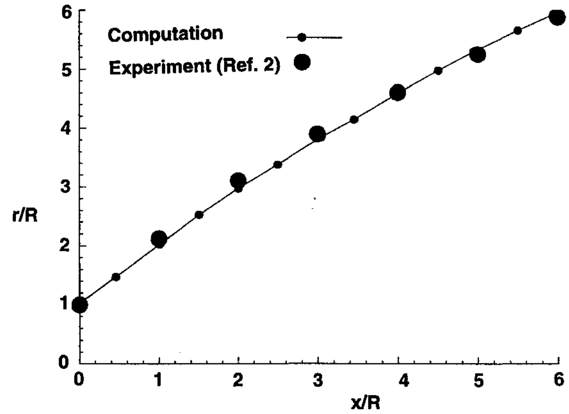

Fig. 3 Plume shape: $M_\infty=4$, $M_j=3$, and $P_j/P_\infty=1500$.

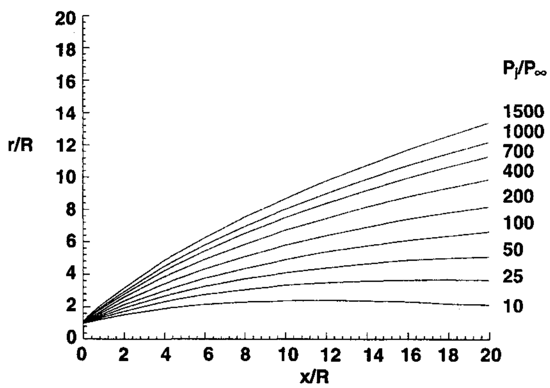

Fig. 4 Effect of pressure ratio on plume shape: $M_\infty=4$ and $M_j=3$.

Conclusion

An axisymmetric Euler solution technique was used to predict the jet plume and freestream interaction. Triangular meshes were used with suitable adaptation techniques. Plume boundary and other flowfield details were obtained. Prediction of jet plume boundary was in good agreement with experimental results. Accuracy depends strongly on the initial and adapted meshes. The Euler solution provides flowfield details that may not easily be obtained experimentally and cannot be predicted with simplified methods. The cost of this solution may be considered low for a computational technique. A Navier–Stokes solution that takes into account the different specific heat ratios for jet and external flow may be worthwhile for a better understanding of the flow structure.

Acknowledgments

The authors would like to thank Özgür Ertunç and Serdar Dilaver for their help. This work was presented as Paper ICAS-96-7-10-4 at the 20th International Congress of the Aeronautical Sciences, Sorrento, Italy, Sept. 8–13, 1996.

References

[1]Nash, K. L., Whitaker, K. W., and Freeman, L. M., "Predicting Exhaust Plume Boundaries with Supersonic External Flow," *Journal of Spacecraft and Rockets*, Vol. 31, No. 5, 1994, pp. 773–777.

[2]Henson, J. R., and Robertson, J. E., "Methods for Approximating Inviscid Jet Boundaries for Highly Underexpanded Supersonic Nozzles," Arnold Engineering Development Center, AEDC-TN-67-2, Tullahoma, TN, May 1962.

[3]Latvala, E. K., and Anderson, T. P., "Experimental Determination of Jet Spreading from Supersonic Nozzles at High Altitudes," Arnold Engineering Development Center, AEDC-TN-58-98, Tullahoma, TN, Nov. 1958.

[4]Love, E. S., and Lee, L. P., "Shape of Initial Portion of Supersonic Axisymmetric Free Jets at Large Pressure Ratios," NACA TN-4195, Jan. 1958.

[5]Latvala, E. K., "Spreading of Rocket Exhaust Jets at High Altitudes," Arnold Engineering Development Center, AEDC-TR-59-11, Tullahoma, TN, June 1959.

[6]Weinacht, P., and Sahu, J., "Navier Stokes Predictions of Missile Aerodynamics," AGARD Rept. 804, Special Course on Missile Aerodynamics, von Kármán Inst., Rhode Saint Genèse, Belgium, and Middle East Technical Univ., Ankara, Turkey, June 1994.

[7]Yılmaz, E., "Two Dimensional and Axisymmetric Euler Solution on Unstructured Grids," M.S. Thesis, Dept. of Aeronautical Engineering, Middle East Technical Univ., Ankara, Turkey, Dec. 1994.

[8]Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solution of Euler Equations by Finite Volume Methods Using Runge Kutta Time Stepping Schemes," AIAA Paper 81-1259, 1981.

[9]Mavripilis, D., "Solution of Two Dimensional Euler Equations on Unstructured Triangular Meshes," Ph.D. Thesis, Dept. of Mechanical and Aerospace Engineering, Princeton Univ., Princeton, NJ, June 1987.

R. M. Cummings
Associate Editor

Evaluation of Engineering Heat Transfer Prediction Methods in High-Enthalpy Flow Conditions

G. Simeonides,* L. Walpot,† M. Netterfield,‡ and G. Tumino§
European Space Agency,
2200 AG Noordwijk, The Netherlands

Nomenclature

C = constant, f^n (geometry, nature of boundary layer)
h = enthalpy, J/kg
n = exponent, 0.5 for laminar flow and 0.2 for turbulent flow
Pr = Prandtl number
$\dot{q}$ = convective heat transfer rate, W/m^2
u = velocity, m/s
u_{grad} = velocity gradient, u_e/x in planar/conical flow
θ = momentum thickness, m
μ = viscosity, Ns/m^2
ρ = density, kg/m^3

Subscripts

ad.wall = adiabatic wall/recovery
main = boundary-layer edge (in main flow direction)

Superscript

$*$ = reference enthalpy and local pressure

Presented as Paper 96-1860 at the AIAA 31st Thermophysics Conference, New Orleans, LA, June 17–20, 1996; received July 8, 1996; revision received Aug. 29, 1997; accepted for publication Nov. 6, 1997. Copyright © 1997 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

*Research Fellow, Aerothermodynamics Section (YPA), European Space Research and Technology Center; also Research Fellow, Department of Aeronautics/Aerospace, von Kármán Institute, 1640 Rhode St-Genèse, Belgium; currently Research Fellow, Laboratory of Thermal Turbomachines, Fluids Section, National Technical University of Athens, P.O. Box 64069, 15710 Athens, Greece.

†Research Assistant, Aerothermodynamics Section (YPA), European Space Research and Technology Center, P.O. Box 299; also Research Assistant, Aerospace Engineering Department, Delft University of Technology, P.O. Box 5058, 2600 Delft, The Netherlands.

‡Researcher, Aerothermodynamics Section (YPA), European Space Research and Technology Center, P.O. Box 299; also Researcher, Fluid Gravity Engineering, Chiltlee Manor, Haslemere Road, Liphook, Hampshire GU30 7AZ, England, United Kingdom.

§Research Assistant, Aerothermodynamics Section (YPA), European Space Research and Technology Center, P.O. Box 299; also Research Assistant, Dipartimento di Meccanica, University of Rome "La Sapienza," via Eudossiana 18, Rome, Italy.

Introduction

APPROXIMATE heat transfer formulations, based on the reference enthalpy extension of boundary-layer solutions over attached flow regions, are applied to generic configurations in the high-enthalpy flow regime. The boundary-layer edge conditions are provided by Euler computations accounting for thermochemical nonequilibrium effects. Comparison of the coupled Euler/reference enthalpy results to thermochemical nonequilibrium Navier–Stokes predictions and to experimental data from hypervelocity/high-enthalpy wind tunnels shows that the proposed methodology gives reasonable engineering heat transfer estimates in the high-enthalpy, thermochemically active flow regime at a significantly lower cost than that of Navier–Stokes computations.

Prediction Methodology

The heat transfer prediction methodology employed involves analytical expressions for convective heating over stagnation regions, infinite swept leading edges, and planar and conical surfaces, based on (or consistent with) Eckert's[1] reference enthalpy concept. These expressions[2] have been generalized into a unique formulation[3]:

$$\dot{q} = CPr^{-\frac{2}{3}}(\rho^*)^{(1-n)}(\mu^*)^n u_{\text{main}}^{(1-2n)} u_{\text{grad}}^n (h_{\text{ad.wall}} - h_{\text{wall}}) \tag{1}$$

A well-identified main flow direction is required; therefore, the configuration must be approximated by a series of generic two-dimensional or axisymmetric aerodynamic surfaces. The boundary layer is treated in thermochemical equilibrium; in particular, the reference density and viscosity are computed from the local pressure and reference enthalpy on the basis of a thermochemical equilibrium assumption, and so is the wall enthalpy. This procedure is aimed to provide reasonable estimates for either equilibrium flows or nonequilibrium flows over fully catalytic walls, i.e., with equilibrium wall conditions. Special attention is drawn to the nonapplicability of analytical expressions for the viscosity of air (such as Sutherland's or Keyes' laws) at high temperatures.[4]

Moreover, the formulation of Eq. (1) does not account for abrupt changes in the boundary-layer growth rate caused either by changes in the geometry (and, thus, in the boundary-layer edge conditions) or by the occurrence of laminar–turbulent transition. A typical criterion for matching transitions between different flow situations (with different boundary-layer growth rates) is the continuity of the momentum deficit,[5] $\rho_{\text{main}} u_{\text{main}}^2 \theta$, through which the effective origin of boundary layers over consecutive parts of the vehicle (flow situations) may be determined by appropriate expressions.[2,3]

The required boundary-layer edge conditions are provided by the Euler version of the TINA solver,[6] which accounts for thermochemical nonequilibrium effects but not for any viscous interaction involving a significant modification of the effective body shape by the growth of dominant thick boundary layers. Similarly, regions of shock wave/boundary-layer interaction and significant