[3]Bishop, R. H., and Azimov, D. M., "Analytical Trajectories of an Extremal Motion with Low-Thrust Exhaust-Modulated Propulsion," *Journal of Guidance, Control, and Dynamics*, Vol. 38, No. 6, 2001, pp. 897–903.

[4]Azizov, A. G., and Korshunova, N. A., "On an Analytical Solution of the Optimum Trajectory Problem in a Gravitation Field," *Celestial Mechanics*, Vol. 38, No. 4, 1986, pp. 297–306.

[5]Azimov, D. M., and Bishop, R. H., "Extremal Rocket Motion with Maximum Thrust in a Linear Central Field," *Journal of Spacecraft and Rockets*, Vol. 38, No. 5, 2001, pp. 765–776.

[6]Tapley, B. D., "Regularization and the Computation of Optimal Trajectories," *Celestial Mechanics*, Vol. 2, No. 3, 1970, pp. 319–333.

[7]McCue, G. A., "Quazilinearization Determination of Optimum Finite-Thrust Orbital Transfers," *AIAA Journal*, Vol. 5, No. 4, 1967, pp. 755–763.

[8]Enright, P. J., and Conway, B. A., "Optimal Finite-Thrust Spacecraft Trajectories Using Collocation and Nonlinear Programming," *Journal of Guidance, Control, and Dynamics*, Vol. 14, No. 5, 1991, pp. 981–985.

[9]Kornhauser, A. L., Lion, P. M., and Hazelrigg, G. A., "An Analytic Solution for Constant-Thrust, Optimal Cost, Minimum-Propellant Space Trajectories," *AIAA Journal*, Vol. 9, No. 7, 1971, pp. 1234–1239.

[10]Novoselov, V. S., *Analytical Theory of Trajectory Optimization in the Gravitational Fields*, Leningrad State Univ. Press, Leningrad, U.S.S.R., 1972, p. 317.

[11]Azizov, A. G., and Korshunova, N. A., "On Optimal Trajectories in Gravitational Fields, Admitting Approximation by Central Linear," *Space Research*, Vol. 29, No. 4, 1991, pp. 525–531.

[12]Jezewski, D. E., "Optimal Analytic Multi-Burn Trajectories," *AIAA Journal*, Vol. 10, No. 5, 1972, pp. 680–685.

[13]Szebehely, V. G., *Adventures in Celestial Mechanics*, Univ. of Texas Press, Austin, TX, 1993, pp. 56–81.

D. L. Edwards
Associate Editor

Performance Analysis of Thermal Protection System of a Solid Rocket Nozzle

V. Jones* and K. N. Shukla†
Vikram Sarabhai Space Center, Trivandrum 695 022, India

Introduction

ABLATIVE materials are commonly used to protect the solid rocket nozzle walls exposed to high-temperature gaseous combustion products. An accurate prediction of the thermal response of these materials is essential for a nozzle thermal designer to successfully carry out the design of an optimum thermal protection system (TPS). Numerous test results on heat transfer in sea-level nozzles are available. Therefore, it is not too difficult for a nozzle thermal designer to design an optimum TPS in the case of sea-level nozzles. However, test results for high-altitude nozzles are very scarce. Consequently, a considerable degree of uncertainty exists in the estimation of wall heat flux, as well as the design of a TPS for a high-altitude rocket nozzle. The most widely used approach for computing wall heat flux in a rocket nozzle is the boundary-layer method developed by Elliott et al.[1] It is always desirable to perform a detailed study in order to supplement the existing design code and validate it with realistic test results so as to design an optimum TPS for the high-altitude rocket nozzles. The present investigation couples two different computer codes previously developed by these authors: a computational fluid dynamics (CFD) code to simulate the fluid flow inside a rocket nozzle[2] and a material thermal response code to study the in-depth response of TPS materials exposed to the hot gas flowing through the rocket nozzle.[3] These two codes are explicitly coupled through an energy balance at the common wall boundary of the nozzle. This Note presents in detail the coupling technique[4] of the two codes and compares the computed results with the test results generated in house as well as those available in the literature.

Received 13 June 2002; revision received 12 January 2003; accepted for publication 30 January 2003.

*Scientist.

†Scientist. Associate Fellow AIAA.

Description of the Problem

In a solid rocket nozzle, normally the inner wall is formed with an ablative liner material such as carbon phenolic or silica phenolic, followed by its structural backup. While the rocket motor is in operation, the liner is subjected to the flow of hot gaseous combustion products, as a result heat flows from the hot gas into the liner material. The ablative liner absorbs the heat and protects the underlying structure by keeping its temperature within tolerable limits. Thus, it is essential for a nozzle thermal designer to compute accurately the heat flow rate from the hot gas into the liner material and to predict accurately the response of charring ablators exposed to the hot gas flow of combustion products.

During the days when the high-speed computing systems were yet to be invented, a closed-form equation,[5] which could be hand computed, was used for estimating nozzle wall heat flux. With the availability of high-speed computing systems, a more sophisticated solution[1] could be adopted for heat-flux computation. However, for high-altitude rocket nozzles it is always desirable to carry out a detailed study by solving Navier–Stokes (N-S) equations for computing nozzle wall heat flux and validate it with realistic test results so as to design an optimum TPS. Such a detailed analysis has now been possible, and the same has been dealt with in the following sections.

CFD in Rocket Nozzles

Jones and Shukla[2] described an analysis of the flow in a rocket nozzle and the development of the associated computer code. The steady viscous turbulent compressible flow in a converging–diverging axisymmetric nozzle is simulated through a computer code using a time-marching explicit scheme. The N-S equations governing an axisymmetric flow for the physical domain of a rocket nozzle are transformed to a rectangular computational domain with a boundary-fitted coordinate system. The effect of turbulence is incorporated in the code by using the Baldwin–Lomax model. The equations are cast into finite difference form in a variable mesh network. The functional values at the interior mesh points are computed using MacCormack's explicit predictor–corrector finite difference scheme; a two-step characteristic scheme is adopted for applying boundary conditions using two independent variables.

In addition to the computation of the usual flowfield variables such as density, pressure, velocity, temperature, etc., the momentum and energy thicknesses are also computed using the respective integrals, for the estimation of heat flux to the wall. The convective heat flux to the wall q_w is computed as follows:

$$q_w = C_h \rho U c_p (T_{\mathrm{ad}} - T_w) \tag{1}$$

The expressions for computing other parameters such as skin friction C_f, Stanton number C_h, etc., are given by Bartz.[6] Boundary-layer interaction exponent $n = 0.1$ and C_f for film property conditions are assumed.

In-Depth Response of Wall Materials

The nozzle wall is assumed to consist of a charring ablator followed by a noncharring structural backup. An analysis and the development of a computer code to predict the in-depth response of charring ablators exposed to high-temperature environments is available elsewhere.[3] The governing mathematical equations are derived in a fixed coordinate system tied to the original surface. The adoption of a fixed coordinate system eliminates the use of a convection component, which is inherent in a moving coordinate system while giving the same results as that of the latter. In-depth pyrolysis kinetics is assumed to obey an Arrhenius law in a reaction zone, and the thermophysical properties are treated as temperature as well as state (virgin, partially charred, or fully charred) dependent. These equations are transformed into finite difference form implicit in time, and a computer code is developed for solution. The parameters such as erosion, char depth, charring ablator/structural backup interface temperature, in-depth density, and temperature fields, etc., can be estimated by this code.

Coupling of CFD and Material Response Codes

The governing equations describing the conservation of mass, momentum, and energy for fluid flow in the rocket nozzle and the equations of mass and energy to characterize the thermal response of the charring wall material and the development of respective computer codes were discussed earlier. The two codes are coupled along the active surface of the nozzle wall by the following energy balance:

$$q_{\text{cond}} = q_w + q_{\text{rad}} - \dot{m}_g \Delta H_g - \dot{m}_c \Delta H_c \tag{2}$$

where q_{cond} is the energy conducted into the material, q_w is the convective heat flux to the wall obtained from the CFD code, and q_{rad} is the radiative heat exchange. The third and fourth terms on the right-hand side represent the energy convected away by the pyrolysis gases passing through the surface and the energy consumed for surface char ablation, respectively. The surface char ablation is assumed to take place once the temperature of the charred surface reaches a particular temperature T_{ab}, known as the char ablation temperature, whose value varies from material to material.

The coupling is done explicitly such that the convective heat flux q_w is computed from the CFD code, using Eq. (1), at each boundary node of the nozzle wall for the initial chamber pressure and the ambient surface temperature of the wall. The computed heat flux is fed into the thermal response code, and this code is time marched at each nodal point for a fixed time step. The new surface temperature of the nodal points and the chamber pressure for the corresponding time, taken from the pressure vs time arrays of the predicted motor performance, are then input to the CFD code. The process is repeated for each time step until the end of motor operation. CPU time is considerably reduced by assuming the CFD solution as the initial condition for the subsequent time step of the duration of motor operation.

Alternatively a matrix of $q_w(T_w, \varepsilon)$ is generated by the CFD code for the average chamber pressure, where ε is the area ratio of the section of the nozzle, which varies from the nozzle inlet to nozzle exit, and T_w varies from ambient temperature to the char ablation temperature of the material. Once this matrix is generated, the thermal response code need not invoke CFD code for obtaining q_w. Instead, it can scan the $q_w(T_w, \varepsilon)$ matrix and obtain the required q_w. This procedure has further reduced the CPU time drastically.

Validation of the Code

The modules developed and presented are based on a detailed CFD analysis of fluid flow in rocket nozzles and heat-transfer analysis of ablative wall materials. The validity of the analysis and accuracy of the final code can be established by comparing the computed results with the test results. Two different sample problems for which test results are available are presented. In the CFD module a convergence tolerance of 0.0003% is specified on the axial velocity in the expansion region. The computations are performed on an IBM RS/6000-44P-270 system.

Sample Problem 1

Consider the measured heat flux into the wall of a conical nozzle of exit area ratio, $\varepsilon = 20.0$ and as reported in Ref. 1. The motor was operated with a N_2O_4–N_2H_4 combination at a chamber pressure of 301 psia. Other nominal conditions are assumed to be the same as reported in Ref. 1. The computational domain (nozzle region only) is divided into 45 × 21 mesh points. The solution converges in 2644 iterations and takes 14.2 s, CPU time. As shown in Fig. 1, nozzle wall heat flux computed by the CFD code compares well with the test results.

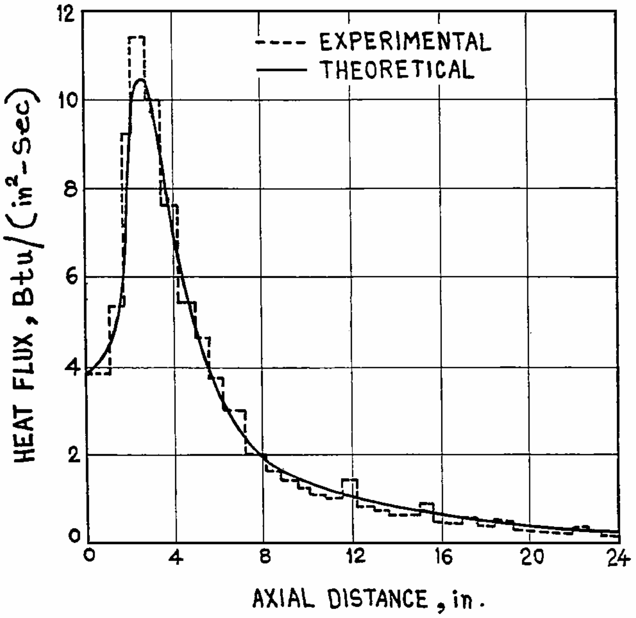

Fig. 1 Comparison of heat flux for sample problem 1.

Sample Problem 2

The static test of a solid rocket motor tested under high-altitude conditions is described elsewhere.[7] The motor has a contour nozzle with throat diameter of 8.0 cm and expansion ratio of $\varepsilon = 50.0$. The throat insert extending to $\varepsilon = 2.7$ is made of graphite. Carbon phenolic is provided as liner for the expansion ratio $\varepsilon = 2.7$–13.9, and thereafter it is silica phenolic. Both the composites are tape wound, parallel to the nozzle axis. The motor operated for 15.5 s with an average pressure of 4.78 MPa. The pressure vs time trace recorded from the static test is reported in Ref. 7. Properties of the combustion products are as follows: $\gamma = 1.19$, $\bar{m} = 29.4$, $c_p = 2198$ J/(kg − K), $T_0 = 3414$ K, and $c^* = 1469$ m/s.

The computational domain of the preceding nozzle is divided into 54 × 24 mesh points. For the computation of cold wall heat flux, the CFD solution converges in 2348 iterations and takes 28.1 s, CPU time. For the generation of $q_w(T_w, \varepsilon)$ matrix, q_w is computed for 21 values of T_w with $T_w = 305, 400, 500, \ldots, 2100, 2173, 2273$ K, where 305 K is the ambient temperature and 2173 and 2273 K are assumed to be the ablation temperatures of carbon phenolic and silica phenolic, respectively.

The coupled code is to be operated for the performance analysis of the TPS. The liner thickness of the nozzle varies from 12.0 mm at $\varepsilon = 2.7$ to 7.0 mm at $\varepsilon = 50.0$. The liner material is divided into nodes of 0.2 mm, and each node is subdivided into eight nodelets. Behind the liner material is a structural backup of fiberglass material of uniform thickness 2.0 mm. The backup is divided into nodes of 0.25 mm and is assumed as noncharring because the system is designed so as to keep the interface temperature around the ambient level only. The time step assumed for this analysis is $\Delta t = 0.02$ s. The CPU time taken by the matrix method is 1.12 s per run, whereas that by the direct use of CFD code is 77.6 s. The computed results by the two different procedures at two typical locations of the nozzle are given in Table 1. The thermal properties of carbon phenolic and silica phenolic are available in Ref. 8. Computed cold wall heat flux for the perceding nozzle is plotted in Fig. 2. Computed char depth values in carbon phenolic and silica phenolic are compared in Fig. 3 with those measured from the static test.

Table 1 Comparison of char depth for sample problem 2

Nozzle diameter, mm	Ablator thickness, mm	Char depth q_w from matrix, mm	Char depth q_w direct from CFD code, mm	Char depth measured average, mm
201	11.18	6.70	6.68	5.90
446	8.41	3.37	3.34	3.10

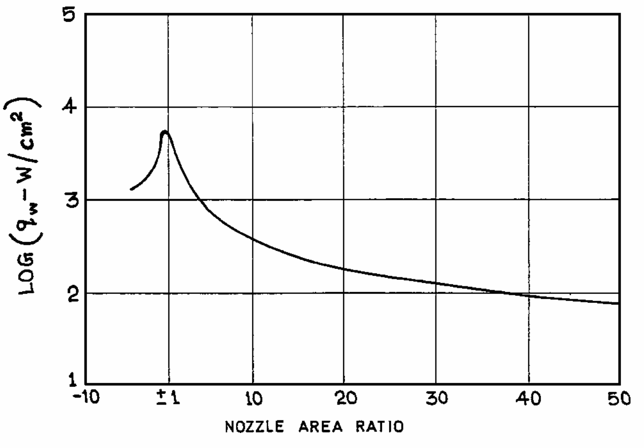

Fig. 2 Computed cold wall heat flux for sample problem 2.

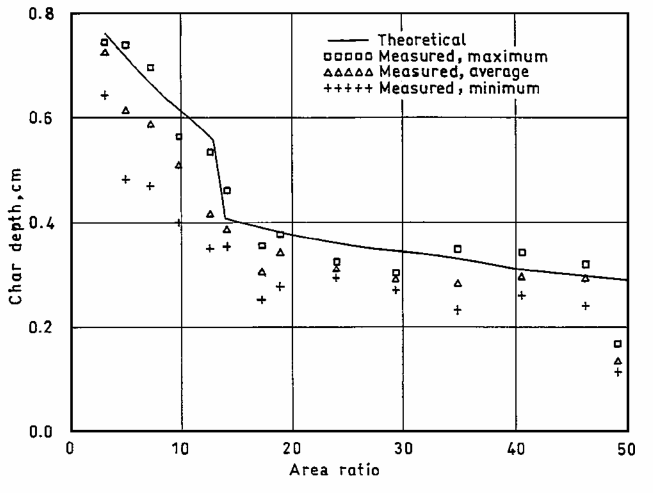

Fig. 3 Comparison of char depth for sample problem 2.

Discussion

Generally, an element of uncertainty exists in the modeling of surface erosion in ablative materials. In the case of sample problem 2, posttest evaluation showed no visible erosion in the nozzle. This is because the motor operated for the duration of 15.5 s only. However, the computations predicted a small erosion of 0.50 mm at the fore end ($\varepsilon = 2.7$) of the nozzle and diminished to zero at $\varepsilon = 5.0$, but still the computed char depth matches the experimental results well. From $\varepsilon = 5.0$ to $\varepsilon = 50.0$, there is absolutely no erosion either experimentally or theoretically. Again as seen from Fig. 3 the measured char depth at $\varepsilon = 49.0$ is much less than that computed. There was some quantity of alumina deposit along the exit region of the nozzle. This alumina shielded the nozzle liner from the exposure to the hot gas and hence reduced the char depth. Thus, this test provides an excellent set of results for comparison with theory, in that it is free from many uncertainties inherent in surface erosion. As seen from Table 1, the computed results obtained by the coupled method are closer to the test results than those of the matrix method. Thermal analysis for a working nozzle in an upper-stage solid rocket motor is also performed. The throat diameter of the nozzle is 16.6 cm and has an expansion ratio at the exit of $\varepsilon = 70.0$. The motor operates for 115.0 s. For the CFD analysis the computational domain is divided into 148×43 mesh points. The matrix method takes 10.0 s for one run, whereas the coupled method takes 300.0 h, CPU time.

Conclusions

An attempt is made to provide a detailed analysis technique for the performance of thermal protection systems of solid rocket nozzles. The present algorithm and the associated computer code are validated by comparing the computed results with test results. This work is aimed at providing a detailed method of analysis to supplement the existing analysis techniques in view of the lack of sufficient test results for high-altitude rocket nozzles, so as to design an optimum TPS having a minimum weight configuration. The algorithm is applicable to sea-level nozzles also. In high-altitude rocket motors the inert weight of the system has a direct or near-direct bearing on the payload of the launch vehicle.

Acknowledgments

The authors express their sincere gratitude to A. K. Verma, E. V. Zoby, T. C. Lin, and the anonymous referees for critically reviewing the manuscript and providing valuable suggestions. The authors are also thankful to S. V. Subba Rao, M. S. Padmanabhan, B. C. Pillai, and M. C. Uttam for their keen interest in the successful completion of this work.

References

[1]Elliott, D. G., Bartz, D. R., and Silver, S., "Calculation of Turbulent Boundary Layer Growth and Heat Transfer in Axisymmetric Nozzles," Jet Propulsion Lab., California Inst. of Technology, TR 32-387, Pasadena, CA, Feb. 1963.

[2]Jones, V., and Shukla, K. N., "Fluid Flow in Rocket Nozzles," AIAA Paper 98-3840, July 1998.

[3]Jones, V., and Shukla, K. N., "In-Depth Response of Charring Ablators to High Temperature Environments," AIAA Paper 99-3462, June 1999.

[4]Jones, V., and Shukla, K. N., "Performance Analysis of Thermal Protection System of a Solid Rocket Nozzle," AIAA Paper 2002-4196, July 2002.

[5]Bartz, D. R., "A Simple Equation for Rapid Estimation of Rocket Nozzle Convective Heat Transfer Coefficient," *Jet Propulsion*, Vol. 27, No. 1, 1957, pp. 49–51.

[6]Bartz, D. R., "Turbulent Boundary-Layer Heat Transfer from Rapidly Accelerating Flow of Rocket Combustion Gases and Heated Air," *Advances in Heat Transfer*, Vol. 2, 1965, pp. 1–108.

[7]Krishnamoorthy, P. A., Pillai, L. A., Padmanabhan, M. S., and Pillai, B. C., "Subscale Performance Simulation Testing for High Altitude Solid Motors," 21st International Symposium on Space Technology and Science, Japan Society for Aeronautical and Space Sciences, Tokyo, May 1998, ISTS Paper 98-a-1-13.

[8]"Testing of Ablative Specimens at Elevated Temperatures," Energy Materials Testing Lab., ME-04005, Final Rept. EMTL-1059, FMI-EMTL-W/A No. 5751, Biddeford, ME, Jan. 1986.

T. C. Lin
Associate Editor