Hypersonic Aerodynamic Heating Prediction Using Weighted Essentially Nonoscillatory Schemes

Theodore K. Lee* and Xiaolin Zhong†
University of California, Los Angeles, Los Angeles, California 90095
Leslie Gong‡
NASA Dryden Flight Research Center, Edwards, California 93523
and
Robert Quinn§
Analytical Services and Materials, Inc., Hampton, Virginia 23666

Introduction

THE calculation of aerodynamic heating rates on hypersonic vehicles remains a challenging problem. Such analysis often involves multidimensional geometries with leading-edge bluntness and fuselage/wing combinations of arbitrary cross sections. In addition, hypersonic flight conditions must take into account the dissociation of air at high temperatures, as well as the resulting effects on surface heating. Numerical simulation of full compressible Navier–Stokes equations for three-dimensional flowfields around these vehicles is still currently too computationally expensive to be routinely used in practical aerodynamic heating calculations.

Because of the large computational time required to compute viscous flows over full aircraft geometries, inviscid flow solutions are computed instead. If an inviscid flow solution is known, then a practical engineering approach for aerodynamic heating calculations is to use a boundary-layer approximation. Current aerodynamic heating methods that are used in this Note for hypersonic flow conditions were developed at NASA Dryden Flight Research Center and are summarized by Quinn and Gong.[1] They provide heating models based on laminar and turbulent flow, using a constant or variable entropy assumption, with the ability to compute transient heating rates as well as steady heating rates.

The motivation for this study is the need for accurate prediction of aerodynamic heating around hypersonic vehicles. NASA Dryden Flight Research Center is interested in extending their hypersonic heating calculation code with advanced computational fluid dynamics (CFD) inviscid solutions. NASA Dryden Flight Research Center has the requirement to conduct flight research on high-speed aircraft. To accomplish this mission and ensure flight safety, it is necessary to have the capability of predicting transient surface heating rates and temperatures.

The accurate calculation of surface heating rates on hypersonic vehicles is dependent on an accurate inviscid CFD numerical solution of the mean aerodynamic flow properties. The shape of the hypersonic vehicle may vary from a simple cone to a complex three-dimensional aircraft. It is because of this constraint that the shock capturing schemes are used. Shock-capturing numerical schemes of up to second-order accuracy have been used in aerodynamic calculations for flows involving shocks.[2,3] Currently, most CFD methods being used in practical applications are second-order accurate at most. The use of numerical methods with a high order of accuracy (third order and higher) is, however, still not prevalent in aerodynamic and heating studies involving supersonic and hypersonic flight. There has not been much implementation of high-order schemes with cases involving realistic aircraft geometries and flight conditions. The shock-capturing method that was implemented was the weighted essentially nonoscillatory (WENO) scheme. The WENO scheme,[4] which is an extension of the ENO scheme,[3,5] is capable of computing robust and high-order-accurate inviscid solutions for hypersonic flows.

CFD Numerical Methods for Inviscid Flow

The accurate calculation of surface heating rates on hypersonic vehicles is dependent on an accurate inviscid CFD numerical solution of the mean aerodynamic flow properties. Shock-capturing numerical schemes have been used in aerodynamic calculations for flows involving shocks.[2,3] The shock-capturing method that was implemented was the WENO scheme. The WENO scheme that is used here is the finite-difference method developed by Jiang and Shu,[4] which is fifth-order accurate. The governing equations for simulating inviscid fluid flows consist of the Euler equations.

There are two air gas models that have been incorporated into the current methods. The first model is a perfect gas model, and the second one is an equilibrium gas model for high-temperature airflows. For flow cases involving temperatures below 2000 K, a perfect air gas model is used. For flow cases involving high temperatures (>2000 K), an equilibrium air gas model is used to account for the dissociation of the O_2 and N_2 molecules and recombination into molecules such as NO. The real gas model uses curve fit formulas for equilibrium air taken from Srinivasan et al.[6] These curve fits are valid up to a temperature of 25,000 K. The method for using the curve fit formulas is described by Grossman and Walters.[7,8]

Aerodynamic Heating Analysis Methods

In the past, the prediction of aerodynamic heating on hypersonic vehicles relied on an inviscid flow solution that was calculated from approximate analytical methods. With the computational methods that are now available, the inviscid flow solutions can be computed with higher accuracy for complex flow conditions, such as angles of attack, complex configurations, complex shock shapes, and real gas effects. Therefore, the solution methodology can begin to apply these new high-order inviscid solutions with the established aerodynamic heating methods. The aerodynamic heating analytical methods that are used in the current work originated from NASA Dryden Flight Research Center's TPATH aeroheating code.[1] These heating methods, however, require a previously computed inviscid flow solution to be used as an input. The TPATH heating methods are able to compute both laminar, turbulent, and transitional flows and flows of constant and variable entropy. The methods are also able to compute steady and transient (trajectory) flows.

Nonstagnation Point Heating

Nonstagnation point heating corresponds to the surface heating along the body away from the stagnation point. There are different methods for both laminar and turbulent flows, as well as constant and variable entropy flows. A constant entropy assumption is based on the properties evaluated at the edge of the boundary layer for the body streamline entropy conditions and weak pressure. For variable entropy assumption, as the boundary layer grows along the length of the body, more of the inviscid flow is entrained into the boundary layer. The streamlines which passed through the bow shock are entrained by the boundary layer, and this entrainment can have a large effect on heating rates.

Constant Entropy Model

One laminar method and two turbulent methods are used. For the laminar method, the method is based on the Blasius incompressible skin-friction formula, which is related to heat transfer with a modified Reynolds analogy. Compressibility effects are accounted

Presented as Paper 2000-0531 at the AIAA 38th Aerospace Sciences Meeting, Reno, NV, 10–13 January 2000; received 13 June 2002; revision received 12 November 2002; accepted for publication 12 November 2002.

*Graduate Student, Mechanical and Aerospace Engineering Department.
†Professor, Mechanical and Aerospace Engineering Department. Associate Fellow AIAA.
‡Aerospace Engineer, Aerostructures Branch.
§Research Scientist, Engineering and Technical Services at NASA Dryden Research Center, 107 Research Drive.

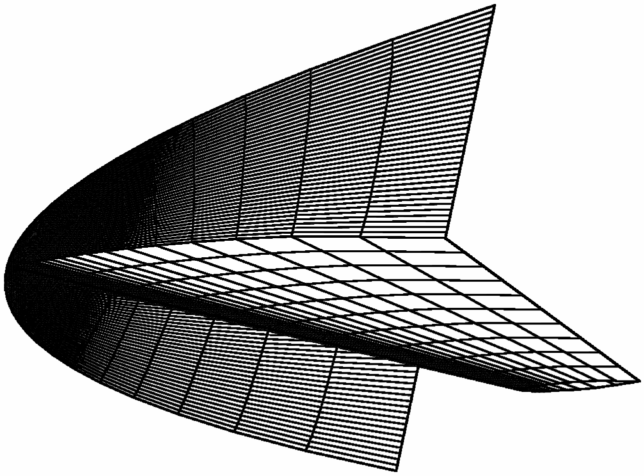


Fig. 1 Example of a wing section grid.

for using Eckert's[9,10] reference enthalpy method. The heat transfer coefficient is

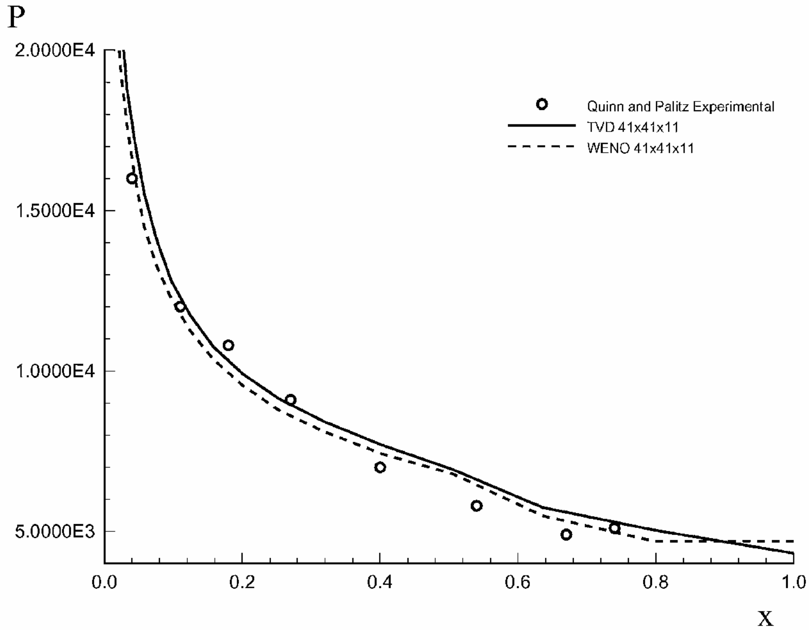
$$h = 0.332F\sqrt{(\rho^* \mu^* V_L/x)}(Pr_W)^{-0.6} \tag{1}$$

where ρ is the density, μ is the viscosity, Pr is the Prandtl number, V is the velocity, the subscript L refers to the wall value, the superscript $*$ refers to reference enthalpy values, and x is the distance from the stagnation point (in feet). $(Pr_W)^{-0.6}$ is the modified Reynolds analogy factor.[1] F is a transformation value[1] with typical values of 1.73 for laminar flows and 1.15 for turbulent flows on cones.

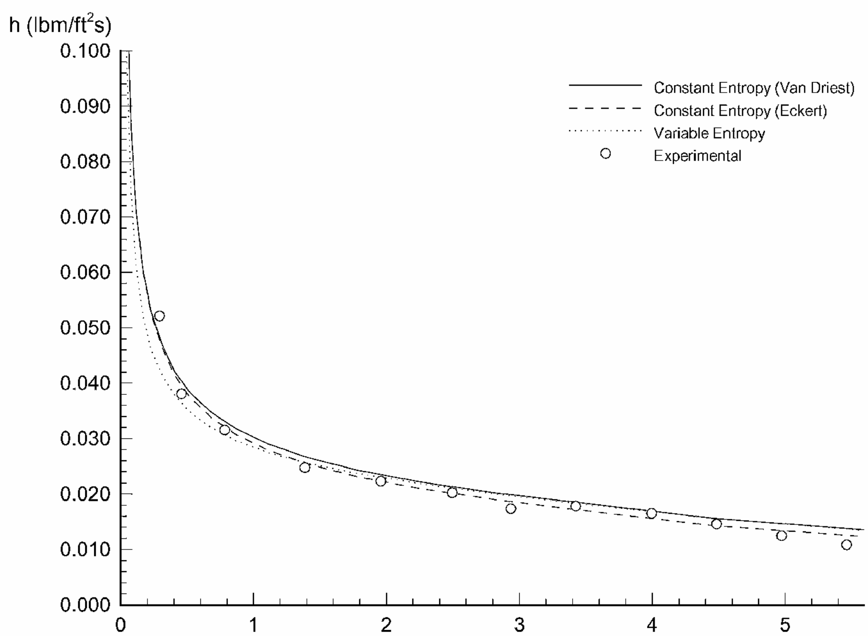
The first turbulent method is based on the skin-friction theory of van driest. The skin-friction coefficient C_f is calculated using the equations found in Ref. 11. The heat transfer coefficient is calculated using

$$h = \frac{FC_f \rho_L V_L}{2Pr_L^{0.4}} \tag{2}$$

The second turbulent method is based on the Schultz–Grunow incompressible skin-friction equation



a)



b)

Fig. 2 X-15 wing lower surface comparisons of experimental values from Quinn and Palitz[13] and two constant entropy methods [Eqs. (2) and (5)] and one variable entropy heating method used with CFD numerical data: a) pressure (pascals) and b) heat transfer coefficient.

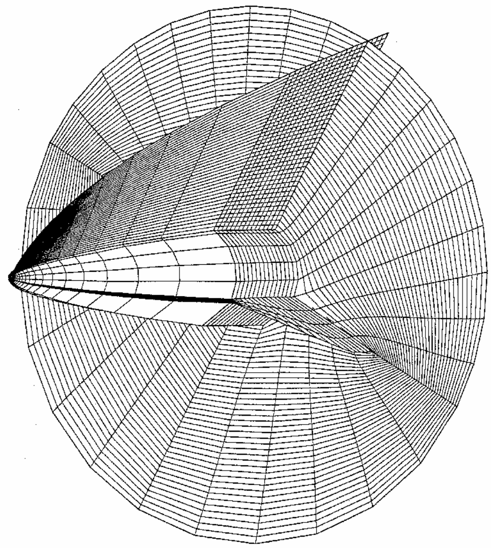


Fig. 3 Cross section of full grid of aircraft geometry similar to X-15.

$$\frac{C_f}{2} = \frac{0.185}{[\log(Re_L)]^{2.584}} \tag{3}$$

where Re is the Reynolds number. The equation is transformed to the compressible flow by Eckert's reference enthalpy method[9,10] resulting in the following equation:

$$\frac{C_f}{2} = \frac{0.185}{[\log(Re_L)]^{2.584}}\left(\frac{\rho^*}{\rho_L}\right) \tag{4}$$

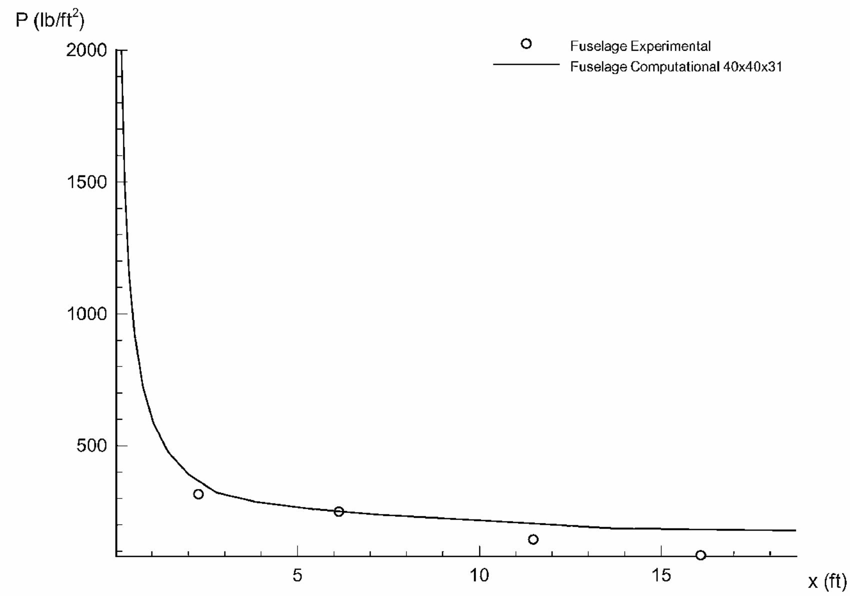
and the heat transfer coefficient is

$$h = F\frac{0.185\rho^* V_L}{[\log(Re^*)]^{4.584}(Pr_W)^{0.4}} \tag{5}$$

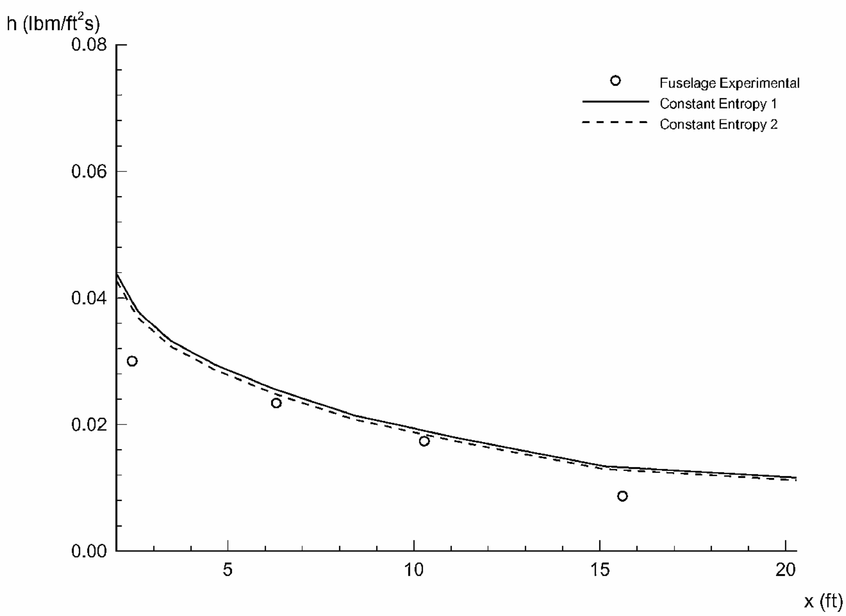
where $(Pr_W)^{0.4}$ is the modified Reynolds analogy factor.

Variable Entropy Model

For variable entropy conditions, the boundary-layer velocity thickness must be considered. The boundary-layer thickness computation procedure is taken from the method described by Zoby et al.[12]



a)



b)

Fig. 4 X-15 fuselage lower surface comparisons of CFD inviscid solution and experimental results of Quinn and Palitz[13]: a) pressure and b) heat transfer coefficient.

For a given boundary-layer velocity thickness, inviscid flow values are found at the boundary-layer edge. The heat transfer coefficient for the laminar variable entropy case is calculated as

$$h = 0.22\left(\rho^*/\rho_L\right)\left(\mu^*/\mu_L\right)(\rho_L V_L/Re_\Theta)(Pr_w)^{-0.6} \qquad (6)$$

where Re_Θ is the Reynolds number based on the momentum thickness. For the heat transfer coefficient for the turbulent variable entropy case, using Eckert's reference enthalpy, the heat transfer coefficient can be calculated using[1,12]

$$h = C_1(\rho_L V_L)(Re_\Theta)^{-m}\left(\mu^*/\mu_L\right)^m\left(\rho^*/\rho_L\right)(Pr_w)^{-0.4} \qquad (7)$$

where C_1 and m are variables described by Zoby et al.[12]

Hypersonic Results

Two grid types were developed and tested with the CFD heating methods. These two grid types are a swept wing and a generic wing–fuselage body.

Heating Calculations for Supersonic Flow over X-15 Swept Wing

The X-15 was a rocket-powered aircraft that was used by NASA to test hypersonic flight conditions in the 1960s. It has a geometry with many three-dimensional effects. There are experimental values of pressure and heat transfer coefficients[13] along the lower surface of the wing, with which a comparison can be made. The numerical grid of the wing is shown in Fig. 1.

Six separate flights of the X-15 that were documented by Quinn and Palitz[13] have been computed that range in Mach number from 4.15 to 5.1. The pressure and heat transfer coefficient comparisons for one flight is shown in Fig. 2. Figure 2 shows the results of a flight at Mach 5.1 with an angle of attack of 2 deg. As can be seen, the heat transfer coefficients that were calculated using the aerodynamic heating program are very close to the experimentally determined heat transfer coefficients.

Three different heat transfer calculations have been used for each flight case. There are two constant entropy calculations based on the analytical models of van Driest[11] [Eq. (2)] and Eckert's reference enthalpy method,[9,10] using Schultz–Grunow's [Eq. (5)] incompressible skin-friction relationship. There is also a variable entropy analytical model [Eq. (7)].

Heating Calculations for Supersonic Flow over X-15 Aircraft

To utilize fully the high-order WENO scheme and the aerodynamic heating methods, more complex three-dimensional geometries needed to be created. A grid over an aircraft meant to simulate the X-15 is shown in Fig. 3. Quinn and Palitz[13] not only documented experimental measurements of pressure and heat transfer coefficient along the lower surface of the wing, but they also documented the measurements of these values along the lower centerline of the fuselage.

The comparison of pressure and heat transfer coefficients along the lower surface of the fuselage is shown in Fig. 4. In Fig. 4, constant entropy 1 method is based on the analytical models of van Driest,[11] whereas constant entropy 2 method is based on Eckert's reference enthalpy method[9,10] using Schultz–Grunow's [Eq. (5)] incompressible skin-friction relationship. Again, Fig. 4 shows a comparison of experimental and computed results for a flight at Mach 5.1 with an angle of attack of 2 deg. The heat transfer coefficients compare closely with the measured values.

Transient Heating of Hypersonic Flow over a Two-Dimensional Wedge

Whereas the preceding heating test cases involved only steady flow, the heating methods are also able to compute surface heating for cases involving flight trajectories. Real situations that hypersonic vehicles will encounter include changing speeds and changing altitudes as well as both laminar and turbulent flow. If the speed of altitude of the hypersonic vehicle changes, then the surface heating will also change with time.

The test case that was chosen was a two-dimensional 10-deg wedge that is 10 ft in length with a nose radius of 0.1 in. This test case was chosen because NASA Dryden Flight Research Center already had results from a previous study.[1] This trajectory profile was provided by NASA Dryden Flight Research Center because it had been used previously in computational work. The range in time for the transient flight was from 40 to 120 s. At 40 s, the freestream conditions are pressure and temperature of 18,754 Pa and 215 K, respectively. The Mach number is 1.9. At 120 s, the freestream conditions are pressure and temperature of 22.6 Pa and 252.5 K, respectively. The Mach number is 6.0.

The following results are based on two locations on the wedge. The first location is at 5.3 ft away from the nose, whereas the second location is at 10 ft away from the nose. The inviscid flow solution was computed using an equilibrium air condition. The surface heat transfer coefficient is shown in Fig. 5. Again, these comparisons are

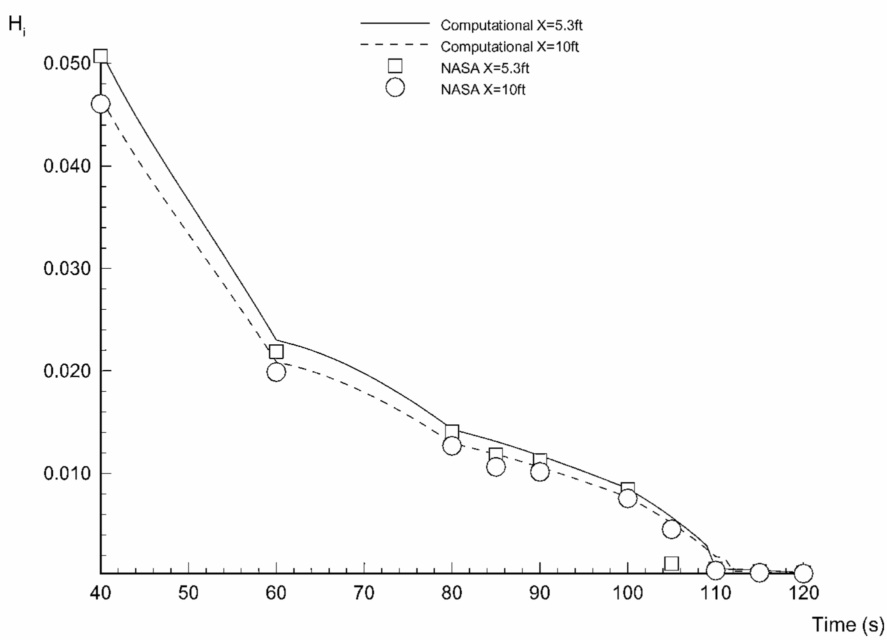


Fig. 5 Heat transfer coefficient (pounds mass per square foot second) comparison between computational solutions and those obtained by Quinn and Gong at two locations along the wedge.[1]

very close to the previous NASA results with a slight delay in the prediction of transition.

Conclusions

The objective of this research was to implement a highly accurate numerical method and combine an analytical heating analysis that can be integrated together to predict heating rates on hypersonic vehicles. The motivation for this research project was the need for accurate prediction of aerodynamic heating around hypersonic vehicles. New grid-generation routines were developed that enable the definition of a generic hypersonic vehicle surface. Complex three-dimensional geometries that include wing and tail sections can now be generated. The WENO high-order numerical scheme is capable of computing both perfect air and equilibrium air flight conditions. The inviscid flow solver incorporates an equilibrium air curve fit formulation to account for flows involving the high temperatures encountered by hypersonic vehicles. The aerodynamic heating methods are capable of computing heating rates for both laminar or turbulent flows. They can use a constant entropy or variable entropy assumption. The heating methods are also capable of computing heating rates for both steady and transient flights.

Acknowledgments

This research was supported by the NASA Dryden Flight Research Center under Grant NCC 2-374. The authors thank Steven Speer of the University of California, Los Angeles, for his help in writing the paper and for producing some of the figures.

References

[1]Quinn, R. D., and Gong, L., "A Method for Calculating Transient Surface Temperatures and Surface Heating Rates for High-Speed Aircraft," NASA TP-2000-209034, Dec. 2000.

[2]Yamamoto, S., and Daiguji, H., "Higher-Order-Accurate Upwind Schemes for Solving the Compressible Euler and Navier–Stokes Equations," *Computers and Fluids*, Vol. 22, No. 2/3, 1993, pp. 259–270.

[3]Shu, C. W., and Osher, S., "Efficient Implementation of Essentially Non-Oscillatory Shock-Capturing Schemes," *Journal of Computational Physics*, Vol. 77, July 1988, pp. 439–471.

[4]Jiang, G. S., and Shu, C. W., "Efficient Implementation of Weighted ENO Schemes," *Journal of Computational Physics*, Vol. 126, No. 1, 1996, pp. 202–228.

[5]Harten, A., Engquist, B., Osher, S., and Chakravarthy, S., "Uniformly High Order Accurate Essentially Non-Oscillatory Schemes III," *Journal of Computational Physics*, Vol. 71, July 1987, pp. 231–303.

[6]Srinivasan, S., Tannehill, J. C., and Weilmuenster, K. J., "Simplified Curve Fits for the Thermodynamic Properties of Equilibrium Air," NASA Reference Publ. 1181, Aug. 1987.

[7]Grossman, B., and Walters, R. W., "Analysis of Flux-Split Algorithms for Euler's Equations with Real Gases," *AIAA Journal*, Vol. 27, No. 5, 1988, pp. 524–531.

[8]Grossman, B., and Walters, R. W., "Flux-Split Algorithms for the Multi-Dimensional Euler's Equations with Real Gases," *Computers and Fluids*, Vol. 17, No. 1, 1989, pp. 99–112.

[9]Eckert, E. R. G., "Survey of Heat Transfer at High Speeds," Wright Air Development Center, TR 54-70, Dayton, OH, April 1954.

[10]Eckert, E. R. G., "Survey of Boundary Layer Heat Transfer at High Velocities and High Temperatures," Wright Air Development Center, TR 59-624, Dayton, OH, April 1960.

[11]van Driest, E. R., "The Problem of Aerodynamic Heating," *Aerospace Engineering Review*, Vol. 15, No. 10, 1956, pp. 26–41.

[12]Zoby, E. V., Moss, J. N., and Sutton, K., "Approximate Convective Heating Equations for Hypersonic Flow," *Journal of Spacecraft and Rockets*, Vol. 18, No. 1, 1981, pp. 64–70.

[13]Quinn, R., and Palitz, M., "Comparison of Measured and Calculated Turbulent Heat Transfer on the X-15 Airplane at Angles of Attack up to 19.0," NASA TM X-1291, Sept. 1966.

W. E. Williamson
Associate Editor

Stagnation-Point Density of Hypersonic Ions in a Repelling Plasma Sheath

Ioannis G. Mikellides* and Myron J. Mandell†
Science Applications International Corporation, San Diego, California 92121
and
Ira Katz‡
Jet Propulsion Laboratory, California Institute of Technology, Pasadena, California 91109

Nomenclature

E	=	specific mechanical energy, J/kg
e	=	electron charge, C
f	=	velocity distribution function, s^3/m^6
$\boldsymbol{h}$	=	specific angular momentum vector, m^2/s
k	=	Boltzmann constant, J/K
M_0	=	Mach number of ions streaming with directed velocity $\boldsymbol{u}_0$
M_∞	=	Mach number of ions at infinity
m	=	ion mass, kg
n_i	=	ion particle density, m^{-3}
n_∞	=	ion particle density at infinity, m^{-3}
$\bar{n}$	=	ratio of local ion particle density to density at infinity
$\bar{n}_{NS}$	=	ratio of stagnation density to density upstream of a normal shock wave
r	=	radius of orbiting ion, m
$\boldsymbol{r}$	=	position vector, m
T_e	=	electron temperature, K
T_i	=	ion temperature, K
$\boldsymbol{u}_0$	=	directed velocity of ions at zero electric potential, m/s
$\boldsymbol{u}_\infty$	=	velocity of ions at infinity, m/s
α_0	=	angle between directed ion velocity and axis of symmetry of hyperbolic orbit, rad
α_∞	=	angle between ion velocity vector at infinity with axis of symmetry of hyperbolic orbit, rad
γ	=	ratio of specific heats
ε	=	eccentricity of ion orbit
θ	=	polar angle in the spherical coordinate system (r, θ, Φ) measured with respect to $\boldsymbol{u}_0$ as polar axis, rad
μ	=	orbit constant, m^3/s^2
Φ	=	azimuthal angle in the spherical coordinate system (r, θ, Φ) measured with respect to $\boldsymbol{u}_0$ as polar axis, rad
φ	=	electric potential, V
χ	=	dimensionless electric potential
ψ	=	angle between ion velocity at infinity and ram ion velocity, rad

Introduction

An important issue in simulating the environment around a low Earth satellite concerns the ram ions that stream toward the spacecraft. Much attention has been given to the wake that forms behind the spacecraft.[1–3] In this Note, we focus on the buildup of

Received 8 January 2002; revision received 16 August 2002; accepted for publication 16 September 2002.

*Senior Staff Scientist, Defense Technology Group, 9455 Towne Center Drive, Mail Stop W2076. Member AIAA.

†Senior Staff Scientist, Defense Technology Group, 10260 Campus Point Drive, Mail Stop X1. Senior Member AIAA.

‡Supervisor, Advanced Propulsion Technology Group, Thermal and Propulsion Engineering Section, 4800 Oak Grove Drive. Senior Member AIAA.