APPLICATION OF CFD TO AEROTHERMAL HEATING PROBLEMS

Michele G. Macaraeg

January 1986
APPLICATION OF CFD TO AEROTHERMAL HEATING PROBLEMS

M. G. Macareeg

NASA Langley Research Center
Hampton, Virginia

Abstract

Numerical solutions of the compressible Navier-Stokes equations by an Alternating Direction Implicit scheme, applied to two experimental investigations at NASA Langley Research Center are presented. The first is cooling by injection of a gas jet through the nose of an ogive-cone, and the second is the aerothermal environment in the gap formed by the wing and elevon section of a test model of the Space Shuttle. The simulations demonstrate that accurate pressure calculations are easily obtained on a coarse grid, with convergence being obtained after the residual reduces by four orders of magnitude. However, accurate heating rates require a fine grid solution, with convergence requiring at least a reduction of six orders of magnitude in the residual. The effect of artificial dissipation on numerical results is also assessed.

1. Introduction

In recent years an increased interest in hypersonic vehicle concepts has brought a number of aerodynamic problems for hypersonic flow into prominence. These problems are characterized by high-temperature high-velocity flows with strong viscous interactions such as shock waves, blunt leading edges, etc. A dominant feature of these flows is the aerodynamically induced high heat transfer rates to the vehicle surfaces. Accurate heat transfer rates are difficult to obtain numerically, and difficult to measure experimentally. However, accurate heating rates are important when designing a given structure and assessing its reliability.

Numerical and experimental models are important tools for understanding the aerothermal environment in high-speed flows. However, before extrapolating the results of such models to actual flight conditions careful model validation along with an understanding of the effects model assumptions have on the simulated physics are important first steps.

This paper will present results from numerical models investigated at NASA Langley Research Center.1 The studies were undertaken to complement experimental results by providing detailed information not obtainable experimentally. In addition, these studies assess the effects artificial dissipation, convergence criteria, and degree of resolution have on pressure and heating rate predictions, and hence provide guidelines for improved techniques. The solution technique is a second-order accurate Beam and Warming Alternating Direction Implicit (ADI) algorithm.2 The algorithm is augmented to include an explicit blend of second and fourth differences for the dissipative terms.3

\begin{equation}
U_x + E_x + F_y + G_z = H(\frac{R_x + S_y + T_z}{Re})
\end{equation}

with

\begin{align*}
U &= \frac{\partial}{\partial x} \\
E &= (\frac{\partial}{\partial x} + \frac{\partial}{\partial y} \frac{E_y}{E}) \\
F &= (\frac{\partial}{\partial x} + \frac{\partial}{\partial z} \frac{F_z}{F}) \\
G &= (\frac{\partial}{\partial y} + \frac{\partial}{\partial z} \frac{G_z}{G}) \\
H &= (\frac{\partial}{\partial y} + \frac{\partial}{\partial z} \frac{H_z}{H}) \\
R &= (\frac{\partial}{\partial y} + \frac{\partial}{\partial z} \frac{R_z}{R}) \\
S &= (\frac{\partial}{\partial y} + \frac{\partial}{\partial z} \frac{S_z}{S}) \\
T &= (\frac{\partial}{\partial y} + \frac{\partial}{\partial z} \frac{T_z}{T})
\end{align*}

where $U, E, F, H, R, S,$ and $T$ are the following four-component vectors:
waves and lesser dissipation in the remainder of dissipative terms regulated by the pressure gradient in the local flow field. The algorithm is augmented to include an explicit gradient in the local flow field. Their addition controls the odd-even uncoupling of grid points typical of central difference schemes, and provides strong dissipation in the region of shock waves and lesser dissipation in the remainder of the field.

Reynolds number, \( Re \), the coefficient of thermal conductivity, \( k \), and the ratio of specific heats, \( \gamma \), are held constant. The coefficient of thermal conductivity and the viscosity coefficient are nondimensionalized with respect to freestream values. The pressure, density, and velocity components are related to the total energy per unit volume, \( e \), by the following equation for an ideal gas:

\[
\exp \left[ \frac{\gamma}{2} \left( u^2 + v^2 \right) \right] = \rho (\gamma - 1) + \rho u^2 + \frac{v^2}{2}
\]

Re represents the Reynolds number. The Prandtl number, \( Pr \), the coefficient of thermal conductivity, \( k \), and the ratio of specific heats, \( \gamma \), are held constant. The coefficient of thermal conductivity and the viscosity coefficient are nondimensionalized with respect to their freestream values. The pressure, density, and velocity components are related to the total energy per unit volume, \( e \), by the following equation for an ideal gas:

\[
\exp \left[ \frac{\gamma}{2} \left( u^2 + v^2 \right) \right] = \rho (\gamma - 1) + \rho u^2 + \frac{v^2}{2}
\]

Numerical results are compared with experimental data obtained in the Langley Research Center 8-Foot High Temperature Tunnel.12 The facility is a hypersonic blow-down wind tunnel that operates at a nominal Mach number of 7, in a test medium of combustion products of methane and air. Numerical results will be compared with two nose configurations on a 12.5° half-angle cone: a 3 inch radius tip (R3) and a 1 inch radius tip on an ogive frustum (R1). Unless otherwise stated the following results and comparisons are for R1, the GJNT configuration without the jet.

**III. Gas Jet Nose-Tip**

Before modelling the GJNT, a series of cases for cones without gas injection were studied to assess the effects that convergence, resolution, and artificial dissipation have on numerical results without the added complications of the gas jet. Freestream conditions are \( Ma = 6.7 \), \( Re_m = 1.4 \times 10^6 \) ft.-1, and \( T_m = 407^\circ R \). Boundary conditions are no-slip at solid surfaces, extrapolation at outflow and a wall temperature of 540°R.

Numerical results are compared with experimental data obtained in the Langley Research Center 8-Foot High Temperature Tunnel.12 The facility is a hypersonic blow-down wind tunnel that operates at a nominal Mach number of 7, in a test medium of combustion products of methane and air. Numerical results will be compared with two nose configurations on a 12.5° half-angle cone: a 3 inch radius tip (R3) and a 1 inch radius tip on an ogive frustum (R1). Unless otherwise stated the following results and comparisons are for R1, the GJNT configuration without the jet.

**III. Gas Jet Nose-Tip**

Before modelling the GJNT, a series of cases for cones without gas injection were studied to assess the effects that convergence, resolution, and artificial dissipation have on numerical results without the added complications of the gas jet. Freestream conditions are \( Ma = 6.7 \), \( Re_m = 1.4 \times 10^6 \) ft.-1, and \( T_m = 407^\circ R \). Boundary conditions are no-slip at solid surfaces, extrapolation at outflow and a wall temperature of 540°R.

**III. Gas Jet Nose-Tip**

Before modelling the GJNT, a series of cases for cones without gas injection were studied to assess the effects that convergence, resolution, and artificial dissipation have on numerical results without the added complications of the gas jet. Freestream conditions are \( Ma = 6.7 \), \( Re_m = 1.4 \times 10^6 \) ft.-1, and \( T_m = 407^\circ R \). Boundary conditions are no-slip at solid surfaces, extrapolation at outflow and a wall temperature of 540°R.

Numerical results are compared with experimental data obtained in the Langley Research Center 8-Foot High Temperature Tunnel.12 The facility is a hypersonic blow-down wind tunnel that operates at a nominal Mach number of 7, in a test medium of combustion products of methane and air. Numerical results will be compared with two nose configurations on a 12.5° half-angle cone: a 3 inch radius tip (R3) and a 1 inch radius tip on an ogive frustum (R1). Unless otherwise stated the following results and comparisons are for R1, the GJNT configuration without the jet.

**III. Gas Jet Nose-Tip**

Before modelling the GJNT, a series of cases for cones without gas injection were studied to assess the effects that convergence, resolution, and artificial dissipation have on numerical results without the added complications of the gas jet. Freestream conditions are \( Ma = 6.7 \), \( Re_m = 1.4 \times 10^6 \) ft.-1, and \( T_m = 407^\circ R \). Boundary conditions are no-slip at solid surfaces, extrapolation at outflow and a wall temperature of 540°R.

Numerical results are compared with experimental data obtained in the Langley Research Center 8-Foot High Temperature Tunnel.12 The facility is a hypersonic blow-down wind tunnel that operates at a nominal Mach number of 7, in a test medium of combustion products of methane and air. Numerical results will be compared with two nose configurations on a 12.5° half-angle cone: a 3 inch radius tip (R3) and a 1 inch radius tip on an ogive frustum (R1). Unless otherwise stated the following results and comparisons are for R1, the GJNT configuration without the jet.

**III. Gas Jet Nose-Tip**

Before modelling the GJNT, a series of cases for cones without gas injection were studied to assess the effects that convergence, resolution, and artificial dissipation have on numerical results without the added complications of the gas jet. Freestream conditions are \( Ma = 6.7 \), \( Re_m = 1.4 \times 10^6 \) ft.-1, and \( T_m = 407^\circ R \). Boundary conditions are no-slip at solid surfaces, extrapolation at outflow and a wall temperature of 540°R.

Numerical results are compared with experimental data obtained in the Langley Research Center 8-Foot High Temperature Tunnel.12 The facility is a hypersonic blow-down wind tunnel that operates at a nominal Mach number of 7, in a test medium of combustion products of methane and air. Numerical results will be compared with two nose configurations on a 12.5° half-angle cone: a 3 inch radius tip (R3) and a 1 inch radius tip on an ogive frustum (R1). Unless otherwise stated the following results and comparisons are for R1, the GJNT configuration without the jet.
three and ten inches and both are seen to fall on the same curve.

The effect of convergence criteria on the pressure is illustrated in Fig. 5 which is a plot of numerically predicted profiles for coarse grids and terminated at three different average global residuals. Stagnation pressures divided by the freestream pressure for residual levels of 10\(^{-4}\), 10\(^{-2}\), and 10\(^{-7}\) are 57.89, 56.78 and 56.77, respectively. A four order of magnitude reduction in the residual differs very little from six and seven orders of magnitude reductions.

Converged solutions of pressure are used to illustrate effects of grid resolution in Fig. 6. The ratio of stagnation pressure over freestream pressure is 56.78 for the coarse grid solution and 60.67 for the fine grid case. Differences in the two solutions for pressure are small, the discrepancy between 1 1/2 and 2 nose radii reflect the inadequacy of the coarser grid in the tangential direction to resolve the expanding flow around the hemispherical nose.

Fig. 7 is a plot of numerically predicted pressures (residual = 10\(^{-6}\)) from a coarse grid solution versus surface distance for different levels of artificial dissipation. The effect of doubling the dissipation has a noticeable effect on the pressure ratio profiles: higher dissipation resulting in lower heating rates is illustrated in Fig. 11. The normalization factor for all bodies in hypersonic flow a study of the cooling effectiveness produced by mass injection was conducted in the 8' HHT. A parallel study is presented using the Navier-Stokes code.

Gas Jet Nose-Tip

To reduce the excessive heating loads on bodies in hypersonic flow a study of the cooling effectiveness produced by mass injection was conducted in the 8' HHT. A parallel study is presented using the Navier-Stokes code. Freestream conditions are \( \rho_{\infty} = 0.67, u_{\infty} = 1.4 \times 10^6 \text{ ft.-l}, \ y = 1.382, \ Prandtl number = .743 \). The gas jet has a temperature of 493°F and is sonic at the outlet. The total pressure of the coolant divided by the pitot pressure of the flow is 2.46 producing an underexpanded jet of coolant gas. These conditions are the same as tunnel test conditions. However, the experiment is in a test medium of combustion products of methane and air and the gas jet is nitrogen, while the numerical model assumes perfect gas relations and ignores molecular diffusion. Numerical results are obtained on a 35 x 41 grid (35 points along the body and 41 normal to the body) and residuals are reduced by 6 orders of magnitude.

In 1965 GJNT experiments conducted by Finley on a blunter body led him to postulate flow features based on Schlieren, shadowgraph and experimental data. A schematic taken from ref. 17 is reproduced in Fig. 13. The schematic depicts a low velocity recirculation zone surrounded by a shear or mixing layer which attaches to the body beyond the recirculation zone. A comparison between GJNT experiments conducted in the 8' HHT and the numerical model reveals similar features.
Fig. 16 is a plot of temperature contours ($T_w = 495 \text{°R}$) which helps illustrate some important flow features. The distance indicated as $s=4.0$ inches (axial) encompasses a recirculation region, as confirmed by numerical values of velocities, where the cold gas is most effective in cooling the body. Downstream of this recirculation zone, at approximately four inches from the nose tip, flow reattaches and the temperature contours move in toward the body though a cold layer of gas still persists down the length of the body. Heating rates remain negative outside the recirculation region, but increase after the recirculation zone. Some of these same features are revealed by image enhancements of experimental photographs. Fig. 17 is an image enhancement of a Schlieren taken of a GJNT experiment. The contrasts help delineate the recirculation region and an outer mixing layer of hot incoming gas mixing with the jet and impinging the body downstream of this recirculation zone. Fig. 18 is a composite of two enhancements of the shadowgraph in Fig. 14. It shows further detail of the recirculating gas jet and depicts a turbulent mixing layer outside the recirculation region. Experimental heat transfer data also indicates turbulent mixing is occurring. These observations are not surprising since the Reynolds number of the gas jet based on the nozzle diameter is approximately $2 \times 10^6$, which is three orders of magnitude higher than the Reynolds number for fully-developed turbulent pipe flow. Fig. 19 is a comparison of numerical and experimental heating rates. These heating rates are nondimensionalized by stagnation point values of a nonblowing experimental case. The heating in the numerical model is due to lateral conduction as expected in a laminar flow. The experimental data show high positive heating rates downstream in the recirculation zone indicating turbulent mixing of the hot incoming gas with the turbulent gas jet. Note that experimental heating rates radially increase at approximately four inches from the nose tip, which is the length of the recirculation region indicated in the temperature contour plot of Fig. 16. However, the numerical heating rates remain negative downstream of the recirculation zone. The numerical results for the adiabatic wall case show wall temperatures below 540°R which confirm the negative heating rates in Fig. 19. Fig. 20 is a plot of adiabatic wall temperatures for a cold gas jet (495°R) and a slightly warmer gas (540°R). The adiabatic temperatures are well below the fixed cold wall temperature, the warmer gas having slightly higher adiabatic temperatures. As expected, adiabatic gas results in higher heating rates as shown in Fig. 21 which plots heating rates for gas jets of 495°R and 800°R. Figs. 20 and 21 reveal that differences in gas jet temperatures are not sufficient to account for the discrepancy between numerical and experimental heat transfer since the increase in heating rates is small for a 395° rise in gas jet temperatures. Thus, the numerical comparison with experiment suggests that maintaining the gas jet as laminar as possible will increase cooling effectiveness.

Comparison of experimental and numerical pressures normalized by stagnation point values of a nonblowing case is given in Fig. 22. The numerical pressures indicate a dip in the recirculating region. Experimental data begin outside of the recirculation zone. Experimental heat transfer data also indicates turbulent mixing is occurring. These observations are not surprising since the Reynolds number of the gas jet based on the nozzle diameter is approximately $2 \times 10^6$, which is three orders of magnitude higher than the Reynolds number for fully-developed turbulent pipe flow. Fig. 19 is a comparison of numerical and experimental heating rates. These heating rates are nondimensionalized by stagnation point values of a nonblowing experimental case. The heating in the numerical model is due to lateral conduction as expected in a laminar flow. The experimental data show high positive heating rates downstream in the recirculation zone indicating turbulent mixing of the hot incoming gas with the turbulent gas jet. Note that experimental heating rates radially increase at approximately four inches from the nose tip, which is the length of the recirculation region indicated in the temperature contour plot of Fig. 16. However, the numerical heating rates remain negative downstream of the recirculation zone. The numerical results for the adiabatic wall case show wall temperatures below 540°R which confirm the negative heating rates in Fig. 19. Fig. 20 is a plot of adiabatic wall temperatures for a cold gas jet (495°R) and a slightly warmer gas (540°R). The adiabatic temperatures are well below the fixed cold wall temperature, the warmer gas having slightly higher adiabatic temperatures. As expected, adiabatic gas results in higher heating rates as shown in Fig. 21 which plots heating rates for gas jets of 495°R and 800°R. Figs. 20 and 21 reveal that differences in gas jet temperatures are not sufficient to account for the discrepancy between numerical and experimental heat transfer since the increase in heating rates is small for a 395° rise in gas jet temperatures. Thus, the numerical comparison with experiment suggests that maintaining the gas jet as laminar as possible will increase cooling effectiveness.

Comparison of experimental and numerical pressures normalized by stagnation point values of a nonblowing case is given in Fig. 22. The numerical pressures indicate a dip in the recirculating region. Experimental data begin outside of this recirculation zone. The comparison indicates good agreement. A calculation of the ratio between the pressure behind the normal shock and $P_{ref}$ for a nonblowing case is 57.702. The highest pressure ratio in the GJNT numerical model, experienced at the lip of the cove, is around 31. Thus the gas jet substantially reduces the pressure load on the nose tip.

**IV. Preliminary Study of a Wing-Elevon Cove**

The last problem to be addressed is a study of the fluid/thermal environment in the cove between the wing and elevon on the Space Shuttle (Fig. 1b). The gap between the Space Shuttle wing and elevon surface is closed by seals at the elevon hinge line to prevent the leakage of hot boundary layer gases. If these hot gases flow into the cove, damage to the thermally unprotected inner wing and elevon structures could occur. To provide insight into the problem a full scale model of the wing-elevon cove was tested in the 8' HTT. A 20 Navier-Stokes solution of the aerothermal environment of the wing elevon cove is given. A major difficulty in obtaining a reliable numerical solution has been producing a computational grid which models the cove geometry adequately. A fully interactive 2D algebraic grid generator currently under development by Erlebacher at Langley, is used to generate the grid required for the present work. Fig. 23a shows the grid produced by this method, for a cove geometry in which the seal "leak" area is equal to the entrance area and the elevon deflection is 25° with respect to the wing. The grid is composed of 55 points on the coordinate family which spans the cove gap, and 81 points on the opposite family. The entrance "neck" region is quite difficult to grid with adequate resolution and near-orthogonality; the enlargement of that region shown in Fig. 23b shows good performance by the above procedure.

In the experimental investigation of the configuration, the elevon/gap apparatus was mounted in a flat test bed inserted into the tunnel; a flat run of approximately 48 inches preceded the cove entrance. The entire test bed was inclined 5° (compression) with respect to freestream. The tunnel freestream conditions were $M_e = 6.7$, $R_e = 1.0 \times 10^6$ ft$^{-1}$ and $T_m = 399$°R; the Reynolds number based on cove entrance gap was 115,000.

Upstream inflow conditions were difficult to set in the numerical model, since the inclination of the experimental apparatus resulted in a weak shock, with a slight entropy layer due to the small radius (3/8") leading edge. Since no detailed surveys of the flowfield upstream of the cove were available, numerical inflow conditions were approximated by setting tunnel freestream conditions inclined at 5° to the wing surface, blending into a flat-plate boundary layer profile near the wing surface. The thickness of the boundary layer was about 20% of the cove entrance gap, consistent with the flat plate run preceding the cove entrance in the experimental apparatus. The effect of these approximate inflow conditions requires further assessment. No-slip, flow tangency and zero normal-pressure gradient were
imposed on the wing and elevon surfaces, along with either adiabatic (zero normal-temperature gradient) or constant wall temperature (540°F) conditions. Extrapolation was used at both freestream outflow and cove "leak" regions.

Fig. 24 is a Mach number contour plot of the flow in the wing-elevon cove. Note the weak compression waves upstream of the cove entrance due to the 5° angle of the input conditions as mentioned previously. Since in the experiment the leading edge is 48 inches from the cove entrance the extrapolation was used and did not have a comparable impact on the flow in the near vicinity of the cove entrance. The boundary layer separates from the wing trailing edge forming a shear layer across the cove entrance which impinges on the deflected elevon downstream where the elevon shock forms. Separation occurs just upstream of the cluster of flow direction arrows at the wing trailing-edge in Fig. 25 indicating a reversal of flow direction. On the cove side of the shear layer a large primary recirculation region forms and flow is directed into the cove. A small secondary recirculating area appears on the elevon surface inside of the cove. In Fig. 25 this secondary vortex is seen as the flow initially heads up the cove and then abruptly reverses direction in a confined region above the recirculation. The primary recirculation is quite strong; local Mach numbers exceed one in some areas. However, little flow is induced farther up in the cove, where Mach numbers are less than .3.

Comparisons are made with experimental data in Fig. 26 for the pressure coefficient C_p versus surface distance. As indicated in Fig. 23a experimental measurements begin at 0 in the cove and proceed down the elevon surface. Numerical values of C_p are higher than experimental data. The prediction of the slope for the steep C_p rise is higher than the experiment. This rise occurs where the shear layer merges with the shock. The occurrence of transition in this separated shear layer is likely to influence the value of this gradient. Experimental results for this test case indicate heat transfer data on the elevon to be within a factor of two of fully turbulent flow as calculated by the referenceenthalpy method, thus suggesting the likelihood that this separated shear layer is indeed in transition. It should be noted that a similar effect might occur if the experimental boundary layer was substantially thicker than the one assumed in this analysis, (the lower velocities causing lower pressure recovery on the elevon and a longer distance before the inviscid level is obtained.)

The value of the C_p (i.e. pressure) plateau on the elevon is within a few percent of the experimental value verifying proper post-shock inviscid conditions in the numerical model. The dip in C_p at approximately 7.5 inches is at the attachment point. There are not enough experimental data points to plot a profile in this section.

VII. Conclusions

Two aerothermal flow problems of relevance in hypersonic flow research have been modelled by compressible Navier-Stokes calculations using an ADI scheme. Comparison with experimental data in both turbulent gas jet. The considerably lower heating rates in the numerical model suggests that maintaining the jet as laminar as possible might greatly enhance its cooling effectiveness.

Preliminary results from numerically modeling a wing-elevon cove indicate good agreement with experimental values of C_p and qualitative flow features. An over prediction by the numerical model in the magnitude of a steep C_p gradient occurring in the vicinity of the separated shear layer may be due to transition effects in the experimental model. Since experimental results indicate heat transfer data on the elevon to be within a factor of two of fully turbulent flow. In addition numerical inflow conditions were approximations to the conditions present in the experimental model; the effect of proper inflow conditions requires further assessment. However, the value of the C_p plateau after its steep rise, predicted by the numerical model is within a few percent of the experimental value verifying the proper post-shock inviscid conditions in the numerical model. Agreement with experimental data in both cases would be greatly enhanced by including turbulence and real gas effects in the numerical model.

For complicated flow fields Beam and Warming is a proven algorithm in performance and reliability. Present day upwind schemes do not yet been tested for complex flow fields, though for simpler problems upwind schemes do exhibit an order of magnitude faster convergence rate than Beam and Warming. However, recent work by Pullian highlights an order of magnitude increase in speed after the addition of implicit order dissipation to the Beam and Warming algorithm, which would make it competitive with upwind methods.
References


Fig. 1. Tests in the 8-Foot High Temperature tunnel at NASA Langley Research Center
a) gas jet nose-tip ogive-cone
b) wing-elevon cove
Fig. 2. Shock Stand-Off Comparison on Nonblowing Ogive-Cone (R1): Numerical and Experimental Results

Fig. 3. Coarse and Fine Grid Solution for R1: Comparison of Shock Stand-Off Distance
a) coarse grid solution
b) fine grid solution
c) magnification of fine grid solution

real gas calculation (ref. 13)
experiment (ref. 4)

Fig. 4. Calculated and Experimental Pressures for Nonblowing Case (R1)

residual = $10^{-4}$
residual = $10^{-6}$
residual = $10^{-7}$

Fig. 5. Effect of Convergence on Pressures (R1)

Coarse grid
Fine grid

Fig. 6. Coarse and Fine Grid Solution of Pressures (R1)
Fig. 7. Effects of Dissipation on Pressure (R1)

Fig. 8. Calculated and Experimental Heating Rates for Nonblowing Cone (R3)

Fig. 9. Calculated and Experimental Heating Rates for Nonblowing Ogive-Cone (R1)

Fig. 10. Effect of Convergence on Heating Rates (R1)

Fig. 11. Coarse and Fine Grid Solution of Heating Rates (R1)

Fig. 12. Effect of Dissipation on Heating Rates (R1)
Fig. 13. Features of a GJNT Flowfield (ref. 17)

Fig. 14. Qualitative Comparison of Numerical and Experimental Models of the GJNT

Fig. 15. Comparison of Numerical and Experimental Shock Stand-Off Distances for GJNT
Fig. 16. Temperature Contour Plot of GJNT

Bow shock
Shear layer
Recirculation zone
Nozzle exit plane

Fig. 17. Image Enhancement of GJNT Schlieren

Fig. 18. Image Enhancement of GJNT Shadowgraph

Fig. 19. Calculated and Experimental Cold Wall Heating Rates for GJNT

Fig. 20. Calculated Adiabatic Temperatures for GJNT

Fig. 21. Effect of Gas Jet Temperatures for GJNT
Fig. 22. Calculated and Experimental Pressures for GJNT

Fig. 23. Grid System for Wing-Elevon Cove Problem
   a) entire computational domain
   b) expanded portion of the "neck" region

Fig. 24. Mach NumberContours of the Wing-Elevon Cove

Fig. 25. Flow Direction Plot in Expanded Portion of the "Neck" Region

Fig. 26. Calculated and Experimental Pressures for the Wing-Elevon Cove
"Application of CFD to Aerothermal Heating Problems"

Michele G. Macaraeg

NASA Langley Research Center
Hampton, Virginia 23665-5225

National Aeronautics and Space Administration
Washington, DC 20546

This paper is to be presented at the 24th AIAA Aerospace Sciences Meeting, January 6-9, 1986/Reno, Nevada.

Numerical solutions of the compressible Navier-Stokes equations by an Alternating Direction Implicit scheme, applied to two experimental investigations at NASA Langley Research Center are presented. The first is cooling by injection of a gas jet through the nose of an ogive-cone, and the second is the aerothermal environment in the gap formed by the wing and elevon section of a test model of the Space Shuttle. The simulations demonstrate that accurate pressure calculations are easily obtained on a coarse grid, with convergence being obtained after the residual reduces by four orders of magnitude. However, accurate heating rates require a fine grid solution, with convergence requiring at least a reduction of six orders of magnitude in the residual. The effect of artificial dissipation on numerical results is also assessed.
End of Document