Boundary Layer Transition
Theoretical Consideration on Laminar-To-Turbulent Transitions over an Ablating Reentry Capsule

By

Kimiya Komurasaki1 and Graham V. Candler2

(1 February 2003)

Abstract: The mechanism of early transition phenomena over an ablating reentry capsule has been examined. A two-equation turbulence model (k – ε model) coupled with Reynolds averaged Navier-Stokes equations has been employed. Low-Reynolds-number effects on the solid wall were considered by modifying the Chien’s correction. As a result, the transition occurred at lower Reynolds numbers with higher ablation rates. The predicted transition-point Reynolds number was $3 \times 10^4$ at the surface-mass-injection rate of 100 g/sm$^2$. The principal mechanism of this early transition is thought as follows; the viscosity damping effects are reduced and re-laminarization is prevented in the downstream of the capsule surface, due to the turbulence on the surface and due to the pushing out of near-surface stream-lines from the surface by successive mass injection.

NOMENCLATURE

- $C_p$ : specific heat at constant pressure
- $d$ : Park’s mixing length on the turbulent wall
- $k$ : turbulence kinetic energy per unit mass
- $K$ : Von Karman constant, 0.41
- $q$ : heating rate on the capsule surface
- $R$ : nose radius, 0.2 m
- $Re$ : Reynolds number, $\rho_\infty V_\infty R/\mu_\infty$
- $Re_{tr}$ : transition point $Re, \rho_\infty V_\infty s_{tr}/\mu_\infty$
- $s$ : distance from stagnation point along the body contour
- $t$ : time
- $T$ : temperature
- $T'$ : turbulence intensity, $T' = \sqrt{2k/3\nu}$
- $u, v$ : mean velocity components
- $u', v'$ : fluctuating velocity components
- $u_r$ : friction velocity, $u_r = \sqrt{\tau_w/\rho}$

1 Department of Aeronautics and Astronautics, University of Tokyo, Tokyo 113-8656, Japan.

2 Department of Aerospace Engineering and Mechanics, University of Minnesota Minneapolis MN 55455, U.S.A.
1. INTRODUCTION

The MUSES-C sample return mission has been proposed by the Institute of Space and Astronautical Sciences of Japan. A spacecraft will rendezvous with a near earth asteroid and bring some samples of the asteroid material back to the earth by a reentry capsule (Park 1998). The spacecraft is scheduled to be launched in 2002 and to return in 2006. The atmospheric reentry speed of the capsule will exceed 12 km/s, and the corresponding reentry Mach number is 42.

The laminar shock layer analyses conducted on this mission indicate that the maximum convective heating rate on the capsule surface would be approximately 10 MW/m² and the radiative rate about 2 MW/m². In order to survive such an extremely severe heating, the reentry capsule will be shielded with a carbon-phenolic ablator. From a material test of the carbon-phenolic ablator using an arc-heater, the recession rate has been estimated at 100 g/sm² at the heating rate of 10 MW/m² (Ishii et al. 1985). This mass-flux corresponds to several % of the free-stream mass-flux. Such a large amount of mass injection from the ablator might induce turbulence over the surface even near the stagnation point of the capsule, resulting in a higher heating rate than predicted by laminar flow analyses (Park & Abe 1984; Park & Balakrishnan 1985; Laurien 1996).

Several experimental studies on the laminar-to-turbulent transitions over a body with surface mass injection have been done in the United States so far, and the results have shown transitions (Demetriades et al. 1976; Kaattari 1979).

Generally speaking, boundary layer transitions hardly occur at a low Re of the order of 10^4, which is a typical MUSES-C reentry condition. However, Kaattari’s experiment predicted the transition at Re = 3 × 10^4 on a blunt body with surface-mass-injection. It implies that transitions might occur during the MUSES-C reentry.

On the ablator surface, the gas injection would be inhomogeneous and unsteady. This surface condition could initiate turbulence in a flow, and eventually cause such a low-Re transition
or an "early" transition (Park & Abe 1984).

The objective of this study is to analytically predict the transition Re and the resulting heating rate distribution over the MUSES-C reentry capsule.

2. EQUATIONS AND PHYSICAL MODELS

2.1 Master equations and Transport coefficients

Reynolds averaged Navier-Stokes equations coupled with the standard $k - \epsilon$ turbulent model (Launder & Spalding 1972) in a cylindrical polar coordinate are solved. Please see the reference (Komurasaki & Candler 2000).

In the first-order turbulent closure model, the Reynolds shear stress $\tau^R$ is expressed through the eddy viscosity, following Boussinesq's assumption:

$$\tau^R_{ij} = \tau^T_{ij} - \frac{2}{3} \rho k \delta_{ij},$$  \hspace{1cm} (1)

where $\tau^T_{ij}$ is the stress component proportional to the eddy viscosity as:

$$\tau^T_{ij} = \mu_T \left( \partial_i u_j + \partial_j u_i - \frac{2}{3} \nabla \cdot \mathbf{u} \delta_{ij} \right) = \frac{\mu_T}{\mu_M} \tau_{ij}.$$  \hspace{1cm} (2)

In the standard $k - \epsilon$ model, the eddy viscosity is expressed as:

$$\mu_T = 0.09 \rho k^2/\epsilon.$$  \hspace{1cm} (3)

Thermal conductivity coefficient can be defined through the Prandtl numbers. Assuming the laminar and turbulent Prandtl numbers for air flows constant at 0.72 and 0.9, respectively, total thermal conductivity is expressed as:

$$\kappa = C_p \left( \mu_M/0.72 + \mu_T/0.90 \right).$$  \hspace{1cm} (4)

In this study, neither chemical reactions nor real gas effects are taken into account, because the boundary transition phenomena is strongly governed by the boundary layer configuration and would not be affected by the location of bow shock nor by the chemical composition. Because of this simplification, calculated post-shock temperature goes up to 80,000 K, which is several times higher than actual post-shock temperature. The molecular viscosity coefficient is obtained using an approximation for non-reacting air with a chemical composition frozen at standard conditions (Hansen 1959).

$$\mu_M = \frac{1.462 \times 10^{-6} T^{1/2}}{1 + 112/T} \text{ kg/ms}.$$  \hspace{1cm} (5)

Recent direct numerical simulations of turbulent reacting flows show that the chemical reactions affect the transition process by either damping or amplifying the disturbances (Martin & Candler 1998). However this effect has not been modeled yet.

2.2 Low-Reynolds-number effects

In order to predict the boundary flow transition, the limiting behavior of the fluctuating velocities approaching a solid boundary has to be considered. This effect is often expressed in
following corrections; 1) "non-isotropic dissipation" of the turbulent energy, and 2) a "damping" effect on eddy viscosity, as proposed by Jones & Launder (1972), Launder & Sharma (1974), Chien (1982), etc.

If the body surface is smooth and non-ablative, the fluctuating velocity satisfies the no-slip boundary condition and also satisfies conservation of mass. Therefore, from expanding the fluctuating velocity in Taylor series near a solid boundary, the tangential and normal components \( u' \) and \( v' \) must behave as follows:

\[
\begin{align*}
u' &= A(s,t) y + O\left(y^2\right), \quad v' = B(s,t) + O\left(y^3\right),
\end{align*}
\]

respectively, as \( y \to 0 \).

The behavior of \( k \) and \( \epsilon \) is deduced from these asymptotic variations of fluctuation velocities as:

\[
\begin{align*}
  k &= \bar{u'}^2 + \bar{v'}^2 = A^2 y^2 + O\left(y^3\right), \\
  \epsilon &= \nu \left(\bar{u'}^2 + \bar{v'}^2\right) = \nu A^2 + O\left(y\right).
\end{align*}
\]

\( k \) decreases rapidly as \( y \to 0 \), while \( \epsilon \) remains finite at \( y = 0 \). This is called as "non-isotropic dissipation" of turbulent energy. To achieve this asymptotic consistency, the "wall" dissipation, \( \epsilon_0 \) is added to the turbulent energy equation in most of the low-Reynolds-number corrections. In Chien's correction, \( \epsilon_0 \) is expressed as:

\[
\epsilon_0 = 2\nu k / y^2.
\]

As for the "damping" effect on eddy viscosity, Eq.(3) is modified using a damping function, \( f_\mu \) as:

\[
\mu_T = 0.09 f_\mu \rho k^2 / \epsilon,
\]

so that \( \mu_T \to 0 \) as \( y^+ \to 0 \). This effect preserves boundary layer flows from transitions until \( Re \sim 10^6 \). In the Chien's correction, \( f_\mu \) depends on a wall variable, \( y^+ \).

\[
 f_\mu = 1 - \exp\left(-0.0115 y^+\right).
\]

However, in the case of a charring ablator such as a carbon-phenolic one, there can exist finite velocity fluctuations even at \( y = 0 \), because the pyrolysis gas is ejecting at subsonic speed through the porous char remnant and allows disturbances to propagate from the exterior flow into the ablator as schematically shown in Fig. 1.

Assuming the limiting case when surface fluctuation velocities perfectly couple with the ones of exterior flow, the wall dissipation, \( \epsilon_0 \) is neglected in this study.

As for the viscosity damping effect, eddy does not vanish on the ablator surface when the injected gas is turbulent. In order to specify the eddy scale on the ablator surface, we define a new wall variable using a non-zero wall variable \( y^+_w \) on the wall as:

\[
y^+ = y (u_T / \nu) + y^+_w.
\]

In this study, two cases are discussed; 1) the eddy scale is larger than boundary layer thickness so that the damping effect is negligible:

\[
y^+_w = \infty.
\]
and 2) the eddy scale is given by the Park’s wall mixing length model (Park & Abe 1984) as:

\[ y_w^+ = \left( d/K \right) \frac{u_T}{\nu}, \]

\[ d = \tau_e v_w. \]

Here \( v_w \) is the mean injection velocity on the surface. A time constant, \( \tau_e \) is an average time interval of fluctuation of injected flow. A reasonable value of the time constant can be taken to be \( \tau_e = 2 \times 10^{-4} \) s for carbon-carbon composite (Park & Abe 1984).

Although some more damping functions are added to the dissipation rate equation in the conventional low-Reynolds-number corrections, they are neglected here because their effects are very small for turbulent-surface conditions.

### 2.3 Free-stream conditions

The free-stream Reynolds number is a key parameter for the amplification or suppression of eddy viscosity in the boundary layer. Reentry conditions at the flight altitude of 60 km and 76 km were tested, corresponding to the free-stream Reynolds numbers of 45000 and 6500, respectively. The conditions are listed in Table 1. (In the MUSES-C mission, peak heating is predicted to be at approximately 60 km of altitude.) The free-stream is assumed turbulence free.

<table>
<thead>
<tr>
<th>Altitude</th>
<th>Density</th>
<th>Temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>76 km</td>
<td>( 3.5 \times 10^{-5} ) kg/m³</td>
<td>195K</td>
</tr>
<tr>
<td></td>
<td></td>
<td>42</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6500</td>
</tr>
<tr>
<td>60 km</td>
<td>( 30.5 \times 10^{-8} ) kg/m³</td>
<td>256K</td>
</tr>
<tr>
<td></td>
<td></td>
<td>38</td>
</tr>
<tr>
<td></td>
<td></td>
<td>45,000</td>
</tr>
</tbody>
</table>
2.4 Ablator surface conditions

The surface is assumed isothermal, and the surface temperature is assumed at 4000 K, which would be the maximum temperature for a carbon-phenolic type ablator. The local mass-injection rates are assumed in proportion to the local heating rates as:

\[ \rho_w u_w = \alpha q. \]  

(15)

The injection-rate can be changed by tuning the coefficient \( \alpha \).

The gas is injected normal to the surface with a given turbulent kinetic energy. The turbulence intensity on the surface is specified to 0.03 - 0.3.

The boundary condition of dissipation rate, \( \epsilon_w \) was given so that \( (0.09 \rho k/c^2)_w = (\mu_m)_w \), while \( \epsilon_w \) doesn’t give any distinguishable effect on the results. The effect of surface roughness has not been considered in this study, though it would be important when the surface mass injection is small.

3. COMPUTATIONAL SCHEME

Flux Vector Splitting method is often used for calculating hypersonic blunt-body flow-fields, because of its robustness and suitability for use in implicit schemes (Candler & MacCormack 1991), while it comes at a price of reduced accuracy due to numerical dissipation.

The numerical dissipation is unfavorable especially to the turbulence calculation, because the turbulence calculation is essentially a calculation of turbulence-dissipation processes.

On the other hand, Flux Difference Splitting method is very accurate, though operation count and complexity are increased for complete linearization of flux formulas for implicit schemes.

Thereby, the AUSM (Advection Upstream Splitting Method) (Liou & Steffen 1993) is employed in this study. This scheme combines the efficiency of FVS and the accuracy of FDS, and enables us to efficiently capture the hypersonic shock and to accurately evaluate boundary layer flows. The Gauss-Seidel line relaxation method (MacCormack 1998) is adopted to implicitly solve the nonlinear equations.

Figure 2 is a plot of the grid used in this study. The master equations were discretized with a third-order upwind scheme using the MUSCL type extrapolation. A grid convergence was obtained for heating rate distributions and for transition positions. A grid convergence has been confirmed: The grid points of 120 in the wall normal direction is found enough to have a convergence in heating rate distribution and the grid points of 30 in the wall tangential direction is enough to have a convergence in transition points.

4. RESULTS

4.1 \( y_w^+ = \infty \) case

The calculated heating rate distributions over the capsule surface are shown in Fig. 3. In the case of \( Re = 45000 \), the stagnation-point heating is decreased with the increase in surface injection rate due to the heat-blockage effect. Every plot shows a peak heating in the downstream region due to the laminar-to-turbulent transition, except for no-injection case. At the injection rate of 80 g/sm², the heating rate profile peaks at \( s = 12 \) cm, and the maximum rate was approximately 25% higher than the stagnation-point one. In the case of \( Re = 6,500 \), the flow stayed laminar for any injection rate.
Figure 4 shows the profiles of mean velocity and turbulence kinetic energy at $Re = 45000$. The plots are enlarged in the surface normal direction. Turbulence kinetic energy is amplified in the stream-wise direction in the middle of boundary layer.

Figure 5 shows the distributions of the eddy viscosity and molecular viscosity. These plots are also enlarged in the surface normal direction. The eddy viscosity surpassed the molecular viscosity in the downstream region.

The peak eddy viscosity along wall normal lines is plotted in the stream-wise direction in Fig. 6. It was normalized by the local molecular viscosity. The nonlinear amplification of the viscosity indicates the transition. The transition point shifts upstream with an increase in the surface-mass-injection rate.

4.2 $y_w^+ = (d/K)(u_r/\nu)$ case

Figure 7 shows the distributions of heating rate and peak eddy viscosity along the surface contour in the case of $y_w^+ = (d/K)(u_r/\nu)$. Comparing with the case of $y_w^+ = \infty$, the transition is delayed due to the viscosity damping effect. The flow stayed laminar at the injection rate less than 80 g/sm$^2$.

Figure 8 shows the heating rate distributions for various injection rates. The transition occurs earlier with larger injection rate.

The distance from the stagnation point to the transition point is plotted for the cases with various surface turbulence intensities in Fig. 9. The figure shows a weak dependence on the surface turbulence intensity. This is due to the fact that the kinetic energy of the injection gas is much smaller than that of main flows and the turbulent energy in the vicinity of surface is
4.3 Injection rate and transition point $Re$

Figure 11 shows a comparison between the Kaattari's transition experiment and the present prediction. The injection rate normalized by the free-stream mass-flux is on the abscissa and determined through the diffusion of turbulent energy from the main flow. Owing to the same reason, the calculation is insensitive to the dissipation rate condition on the surface, $\epsilon_w$. If $T' = 0$ on the surface, the flow stays laminar at any injection rate because there is no seed of turbulence.

Figure 10 shows a streamline of the gas injected near the stagnation point. The seed of turbulence given on the surface is carried along the streamline and is gradually amplified by the work of the main flow against the Reynolds stress. The streamline is separated from the surface and is swelled into the middle of boundary layer by successive mass injection in the downstream region of the surface. At a small injection rate, the streamline stays in the region where the viscosity damping effect is strong, while at a large injection rate, the streamline is pushed into the region where local $Re$ is relatively high and damping effect is weak, resulting in a transition.
Fig. 4: Profiles of mean velocity and turbulent kinetic energy. $Re = 45,000$, $y^+_w = \infty$, $T'' = 0.1$, $(\rho_w v_w)_{sp} = 80 \text{ g/sm}^3$. Enlarged in the wall normal direction by a factor of 5.

Fig. 5: Profiles of eddy viscosity and molecular viscosity. $Re = 45,000$, $y^+_w = \infty$, $T'' = 0.1$, $(\rho_w v_w)_{sp} = 80 \text{ g/sm}^3$. Enlarged in the wall normal direction by a factor of 5.
Fig. 6: Eddy viscosity distribution. $Re = 45,000$, $y_{w}^{+} = \infty$, $T' = 0.1$.

Fig. 7: Distributions of heating rate and eddy viscosity. $Re = 45,000$, $(\rho_{w}v_{w})_{sp} = 80$ gsm$^2$, $T' = 0.1$. 
Fig. 8: Heating rate distributions on an ablator surface. $Re = 45,000$, $y_w^{+} = (d/K)(u_r/\nu)$, $T' = 0.1$.

Fig. 9: Surface turbulence intensity and transition distance from stagnation point. $Re = 45,000$, $y_w^{+} = (d/K)(u_r/\nu)$. 

the transition point $Re$ is on the ordinate.

In the case of $y_0^+ = \infty$, calculated results show earlier transition than the experimental observations. Transitions occur even at the quite small injection rate. That seems quite unrealistic. On the other hand, in the case of $y_0^+ = (d/K)(u_r/u)$, transitions don't occur at the injection rate less than 80 g/sm$^2$. The calculated transition envelope is well agreed with the experimental results.

5. DISCUSSIONS

The early transition would be induced through following two mechanisms: 1) Reduction of the viscosity damping effect due to the surface turbulent condition, and 2) Pushing out of the injected-gas stream-lines toward the middle of boundary layer by successive mass injection in the downstream region of the body.

Both of them assist the amplification of eddy viscosity in the boundary flows. Since the predicted transition point $Re$ shows a good agreement with the Kaattari’s experimental results, it can be concluded that this numerical model well represents the early transition mechanism due to surface-mass-injection.

In the MUSES-C reentry, the increase in heating rate due to the transition would be small so far as the ablation rate is less than 100 g/sm$^2$. 

Fig. 10: A streamline of the gas injected near stagnation point and contours of $(\mu_r/\mu)$. $Re = 45,000$ $y_0^+ = (d/K)(u_r/u)$, $T^* = 0.1$, $(\rho_w v_w)_{sp} = 80$g/sm$^2$. Enlarged in the wall normal direction by a factor of 20.
Fig. 11: Transition point $Re$ and injection rate ratio.

REFERENCES


Park C., 2001, Injection-Induced Turbulence in Stagnation-Point Boundary Layer, AIAA Journal Vol.22,
Anomalous Heating Profile over Entry Capsule with Ablation and Its Relation to Earlier Turbulent Transition

By

Keisuke SAWADA*

(1 February 2003)

Abstract: Effect of earlier turbulent transition and enhancement of turbulence intensity in the boundary layer due to ablation product gas is discussed in terms of anomalous heating profile over the surface of an entry capsule. First, a numerical approach that utilizes Park’s injection induced turbulence model is described and some of the numerical results for code validation study are shown. Second, calculated results of trajectory-based aerodynamic heating environment for Pioneer-Venus entry capsules are shown. Finally, a numerical prediction of trajectory-based aerodynamic heating environment for MUSES-C is made and the possible effect of earlier turbulent transition is discussed.

1. INTRODUCTION

It is customary to employ an ablative heatshield for entry capsule that flies into the atmosphere of a planet with a superorbital velocity. This is because the ablative heatshield has substantially larger heat capacity than that of heat resisting material due to pyrolysis process of phenolic regin, and also to the convective blockage effect of ablation product gas. However, ablation product gas injected from the wall can heavily interact with the shock layer flow. Moreover, the surface of the char layer can be rough due to ablation and surface reactions. Therefore, there is a fear that the otherwise laminar boundary layer can turn to be turbulent which inevitably results in unexpectedly higher aerodynamic heating rate.

In the past, the United States has experienced entry flight into planetary atmosphere with significant ablation in the Pioneer-Venus mission and also in the Galileo mission. From the flight data obtained in the Pioneer-Venus mission, it was shown that the temperature at the thermocouple position embedded in ablator at the downstream frustum region was significantly elevated than that at or near the stagnation point (Wakefield & Pitts 1980). In the Galileo mission, the amount of ablator at the downstream frustum region that was melted during the entry flight was about twice thicker than that given by the preflight prediction (Milos 1996).

* Department of Aeronautics and Space Engineering, Tohoku University, Sendai 980–8579, JAPAN.
It should be noted that the ablative heatshield of Galileo probe vehicle was designed based on the calculation assuming fully turbulent boundary layer from the stagnation point. Despite this assumption, the actual thickness of ablator that melted while the entry flight was far thicker than the prediction. This indicates the importance of identifying the cause of higher heating rate that appeared at the downstream frustum region.

Park (1999) pointed out that the possible causes of higher heating rate observed in the downstream region could be (i) earlier transition of boundary layer and enhancement of turbulence intensity due to ablation product gas, (ii) enhancement of turbulence intensity in both shock layer and boundary layer due to spalled particles, and (iii) nonequilibrium radiation from spalled particles that vaporized in the shock layer. In our previous studies, we focused on the possible turbulent transition and enhancement due to ablation product gas by employing a specially devised turbulence model that gave eddy viscosity of ablation product gas at the wall surface.

In this work, we first give a brief description of this turbulence model that assumes the ablation product gas is inherently turbulent, and then show how this model is implemented into a CFD code, and how this method is validated against the available experimental data obtained in hypersonic wind tunnels. Next, some of the calculated results for the entry flight of Pioneer-Venus probe vehicles are shown together with a comparison of obtained temperature history at the thermocouple positions with actual flight data. Finally, obtained results for MUSES-C entry flight are shown where an emphasis is placed on how the turbulence effect due to ablation product gas alters the aerodynamic heating environment.

2. PARK’S INJECTION INDUCED TURBULENCE MODEL

One can find several related works that investigated earlier turbulent transition of the boundary layer due to ablation product gas. In particular, those experimental studies of Feldhuhn (1976) and Kaattari (1978) are well known. In their works, porous materials were employed for wind tunnel model and pressurized foreign gas was injected through the wall into the boundary layer. When the mass flow rate of foreign gas was increased, turbulent transition occurred at some location from the downstream side, and the corresponding surface heating rate was increased, as shown in Fig. 1. It is interesting to note that the heating rate at the stagnation point decreased only slightly as the mass flow rate was increased. In other words, as shown in Fig. 2, the heating rate at the stagnation point became much higher than that given by the boundary layer theory assuming laminar injection flow.

In order to explain the cause of the observed higher heating rate at the stagnation point, Park (1984) conjectured that the injected gas from the wall was inherently turbulent. He found the functional form of mixing length for injected gas at the stagnation point, and obtained the time constant for each of the porous materials using the available experimental data. According to Park’s model, the eddy viscosity of injected gas at the stagnation point μ_{\text{w}} can be given by

$$\mu_{\text{w}} = 0.4\rho_{\text{w}}\nu_{\text{w}}d,$$

where ρ_w denotes the density of injected gas, υ_w the velocity component normal to the wall, and d the mixing length. The functional form of the mixing length is given by

$$d = d_m\{1 - \exp(-\tau e_{\text{w}}/d_m)\},$$

in which τ is a time constant and d_m the limiting value of the local mixing length. This turbulence model of Park is called as the injection induced turbulence model.
Fig. 1: Surface heating rate ratio for a sphere with surface injection flows (Kaattari 1978). $M_\infty = 7.32$, $Re \approx 10^8$, $P_e = 400\text{psix}$, $T_i \approx 780K$, and $m^* = 0.003(77H1), 0.007(78H1), 0.022(79H1), 0.027(80H1)$ where $m^*$ denotes the mass flow rate ratio.

Fig. 2: Stagnation heating rate ratio for various blowing parameter $B$ in Kaattari’s experiment (Kaattari 1978).

3. IMPLEMENTATION OF TURBULENCE MODEL INTO CFD CODE

In our previous studies, we tried to implement the Park’s injection induced turbulence model into a CFD code by utilizing either one-equation turbulence model (Dendou & Sawada 1998) or zero equation model (Izawa & Sawada 2000a). In this work, we will describe the latter approach utilizing an algebraic turbulence model of Baldwin and Lomax (1978).

A proper form of the wall damping function for injection flow was studied by Cebeci (1970).
The constant $A^+$ in the Van Driest damping function (Van Driest 1956)
\[
D = 1 - \exp\left(-y^+/A^+\right),
\]
was modified, and a favorable agreement with experiment was indicated. However, in the vicinity of the wall, the viscous sublayer region appears, which does not suit with the concept of injection induced wall turbulence modeling. The Van Driest damping function is constructed based on the solution for the incompressible Stokes flow over an oscillating flat plate. The amplitude of oscillation was translated as the fluctuation component of fluid velocity. We adopt the same idea and construct a wall damping function for turbulent wall injection flow. The eddy viscosity used in the calculation is now assumed in the following form as
\[
\mu_t = (\mu_t)_{inj} + (\mu_t)_{BL},
\]
where the first term in the right-hand side is given by
\[
(\mu_t)_{inj} = \rho d_{inj}^2 |\omega|, \quad \text{and} \quad d_{inj} = \max (0, d_w - \chi y) \exp (-y^+/A^+),
\]
in which $\omega$ denotes the vorticity, $\chi$ the Karman constant and is assumed as $0.4$. The mixing length $d_{inj}$ takes the maximum value $d_w$ at the wall and decays exponentially in the boundary layer according to the Van Driest theory. The wall mixing length $d_w$ is chosen to satisfy the relation
\[
(\mu_t)_{Park} = \rho_w d_w^2 |\omega|_w.
\]
The second term in the right-hand side of Eq. (4) is given by the original form of the Baldwin-Lomax turbulence model. Therefore, the eddy viscosity given by this second term vanishes toward the wall.

Let us show the obtained results in the code validation study (Izawa & Sawada 2000a). Figure 2 shows the stagnation point heating rate ratio $\Psi$ for a different blowing parameter $B$ that is the ratio of injected mass rate to the fraction of the maximum available heat arriving at the surface without mass addition (Marvin & Pope 1967). A fair agreement between the calculated stagnation heating rates and the experiments for different Reynolds numbers is shown. Park (1984) calibrated his turbulence model by several available experimental data including those shown in Fig. 2. This agreement therefore comes from the consistency between the Park’s theory and the present treatment of eddy viscosity in the numerical scheme. The polynomial curve fit $\Psi = \Psi(B)$ shown in the figure is obtained from the boundary layer theory, which gives the correlation between the reductions in heat transfer with increasing mass addition parameter at the stagnation point (Marvin & Pope 1967). Note that both the experimental data and the numerical results are substantially larger than the correlated curve for larger blowing parameters.

The calculated surface heat transfer rate ratio is shown in Fig. 3. The heating rate in the downstream region becomes twice as large as the experimental data probably because a simple sum of eddy viscosities given by two different turbulence models in Eq. (4) gives an eddy viscosity that is too large in the downstream region. Note that the location of the local maximum in the heating profile is found to agree well suggesting that the location of turbulent transition is rather well predicted. For comparison, the calculated profile by removing $(\mu_t)_{BL}$ in Eq. (4) is also shown. The heating rate at the stagnation point slightly increases in this case, while it almost monotonically decreases along the wall surface. This demonstrates the importance of turbulent transition caused by the Baldwin-Lomax model for reproducing the experimental data.
4. TRAJECTORY-BASED ANALYSIS OF PIONEER-VENUS ENTRY FLIGHT HEATING ENVIRONMENT

The obtained results of our previous studies (Izawa & Sawada 2000b; Takahashi & Sawada 2002) are shown for indicating how the heating profile along the wall is altered and how well the flight data of temperature histories are reproduced, if we assume the ablation product gas is inherently turbulent.

In the calculations, we solve the Navier-Stokes equations cast in an axisymmetric form. Those 11 chemical species (CO₂, CO, C, C₂, C₃, O, OH, H, H₂O, C⁺ and e⁻) are adopted for the Venusian atmosphere and also for ablation product gas. In order to account for nonequilibrium thermochemical reactions, the Park’s two-temperature model (Park 1989) is employed. Thermal response of ablative heatshield is obtained from the Super Charring Materials Ablation (SCMA) code developed by Ahn et al. (2002). In the SCMA code, four conservation equations for solid density, gas density, gas momentum, and total energy are solved one-dimensionally in the normal direction of the local wall surface. Those CFD and SCMA codes are loosely coupled at the chosen trajectory points for obtaining consistent boundary conditions and trajectory-based heating rates.

In the Pioneer-Venus mission, four probe vehicles entered into the Venusian atmosphere at the same time. In this work, we focus on one of these vehicles, the North Probe, for which the actual entry trajectory was well identified. The entry angle of the North Probe was -68.7 deg and the entry velocity was 11.54 km/s at an altitude of 200 km.

Figure 4 shows the obtained heat flux profiles for the North Probe at 11.5 sec of flight time from 200 km altitude. This trajectory point corresponds to the peak heating point for the North Probe. The heat flux at the stagnation point is decreased from the laminar solution due to convective blockage effect of ablation product gas, but is considerably increased in the downstream region due to turbulence effect. The convergence history of the net heat flux profile along the surface is also shown. It is indicated that the convergence is attained within 5 iterations for this case. At different trajectory points, the convergence is generally attained.
within 5 to 8 iterations. In Fig. 5, the calculated temperature variations at the forward and backward thermocouple positions for the North Probe are compared with the flight data. One can see the calculated temperature profiles agree fairly well with those from the flight data. One can also see the onset of temperature rise in the calculation is slightly earlier than that of the flight data. This is probably due to the clocking error in the flight data (Ahn et al. 2002). Though not shown here, fair agreements in temperature profiles for the remaining probe vehicles were obtained. In particular, the anomalous heating profile at the downstream region was well reproduced (Takahashi & Sawada 2002).

5. TRAJECTORY-BASED ANALYSIS OF MUSES-C ENTRY FLIGHT HEATING ENVIRONMENT

Finally, let us show the obtained results of trajectory-based heating analysis for MUSES-C entry capsule (Suzuki T. et al. 2002a). The same computer code used in the simulation of Pioneer-Venus probe vehicles was utilized, though 21 chemical species for describing air species and also ablation product gas (Air: N, O, N2, O2, NO, N+4, O+, N2+, O2+, NO+, e-, Ablation: C, C2, C3, CO, CN, C+, H, H2, C2H, H+) were employed.

In the calculation, we assumed the entry angle into the atmosphere at an altitude of 200 km was -12 deg. The peak heating occurred at the altitude of 56 km for this case. Figure 6 shows the convective heating profile accounting for ablation and turbulence over the vehicle surface at this altitude. As in the case of Pioneer-Venus probes, the heating rate at the stagnation point slightly decreases from that of the laminar solution (without ablation) due to convective blockage effect, but is increased considerably in the downstream region due to turbulence effect.

The calculated temperature histories at the ablator surface are shown in Fig. 7. At the stagnation region, the obtained temperature history shows a fair agreement with that given by Suzuki et al. (Suzuki K. et al. 1998). They studied the stagnation heating and ablation process.
4. TRAJECTORY-BASED ANALYSIS OF PIONEER-VENUS ENTRY FLIGHT HEATING ENVIRONMENT

The obtained results of our previous studies (Izawa & Sawada 2000b; Takahashi & Sawada 2002) are shown for indicating how the heating profile along the wall is altered and how well the flight data of temperature histories are reproduced, if we assume the ablation product gas is inherently turbulent.

In the calculations, we solve the Navier-Stokes equations cast in an axisymmetric form. Those 11 chemical species (CO₂, CO, C, C₂, C₃, O, OH, H, H₂O, C⁺ and e⁻) are adopted for the Venusian atmosphere and also for ablation product gas. In order to account for nonequilibrium thermochemical reactions, the Park’s two-temperature model (Park 1989) is employed. Thermal response of ablative heatshield is obtained from the Super Charring Materials Ablation (SCMA) code developed by Ahn et al. (2002). In the SCMA code, four conservation equations for solid density, gas density, gas momentum, and total energy are solved one-dimensionally in the normal direction of the local wall surface. Those CFD and SCMA codes are loosely coupled at the chosen trajectory points for obtaining consistent boundary conditions and trajectory-based heating rates.

In the Pioneer-Venus mission, four probe vehicles entered into the Venusian atmosphere at the same time. In this work, we focus on one of these vehicles, the North Probe, for which the actual entry trajectory was well identified. The entry angle of the North Probe was -68.7 deg and the entry velocity was 11.54 km/s at an altitude of 200 km.

Figure 4 shows the obtained heat flux profiles for the North Probe at 11.5 sec of flight time from 200 km altitude. This trajectory point corresponds to the peak heating point for the North Probe. The heat flux at the stagnation point is decreased from the laminar solution due to convective blockage effect of ablation product gas, but is considerably increased in the downstream region due to turbulence effect. The convergence history of the net heat flux profile along the surface is also shown. It is indicated that the convergence is attained within 5 iterations for this case. At different trajectory points, the convergence is generally attained.
within 5 to 8 iterations. In Fig. 5, the calculated temperature variations at the forward and backward thermocouple positions for the North Probe are compared with the flight data. One can see the calculated temperature profiles agree fairly well with those from the flight data. One can also see the onset of temperature rise in the calculation is slightly earlier than that of the flight data. This is probably due to the clocking error in the flight data (Ahn et al. 2002). Though not shown here, fair agreements in temperature profiles for the remaining probe vehicles were obtained. In particular, the anomalous heating profile at the downstream region was well reproduced (Takahashi & Sawada 2002).

5. TRAJECTORY-BASED ANALYSIS OF MUSES-C ENTRY FLIGHT HEATING ENVIRONMENT

Finally, let us show the obtained results of trajectory-based heating analysis for MUSES-C entry capsule (Suzuki T. et al. 2002a). The same computer code used in the simulation of Pioneer-Venus probe vehicles was utilized, though 21 chemical species for describing air species and also ablation product gas (Air: N, O, N₂, O₂, NO, N⁺, O⁺, N₂⁺, O₂⁺, NO⁺, e⁻, Ablation: C, C₂, C₃, CO, CN, C⁺, H, H₂, C₂H, H⁺) were employed.

In the calculation, we assumed the entry angle into the atmosphere at an altitude of 200 km was -12 deg. The peak heating occurred at the altitude of 56 km for this case. Figure 6 shows the convective heating profile accounting for ablation and turbulence over the vehicle surface at this altitude. As in the case of Pioneer-Venus probes, the heating rate at the stagnation point slightly decreases from that of the laminar solution (without ablation) due to convective blockage effect, but is increased considerably in the downstream region due to turbulence effect.

The calculated temperature histories at the ablator surface are shown in Fig. 7. At the stagnation region, the obtained temperature history shows a fair agreement with that given by Suzuki et al. (Suzuki K. et al. 1998). They studied the stagnation heating and ablation process...
Fig. 5: Temperature variations at the thermocouple positions for the North Probe.

Fig. 6: Convective heat flux profiles at 70 sec from 200 km. The results of grid convergence study are also indicated.

of the MUSES-C capsule by using the viscous shock layer equations. Although Suzuki et al. considered the laminar boundary layer with ablation, the heating rate became comparable with the present turbulent solution due to inclusion of catalytic recombination of O and N atoms. At the frustum region, the surface temperature is also elevated significantly, though slightly lower than that at the stagnation point.

The calculated recession profile along the wall surface at 75 sec of flight time is shown in Fig. 8. Because of the significant turbulent heating in the downstream region, the surface
Fig. 7: Wall temperature histories at the stagnation point and in the downstream region of MUSES-C reentry capsule.

recession becomes larger at $s/R = 0.3$ than at the stagnation point and then gradually decreases toward the juncture point but slightly increases again at the frustum region. The amount of surface recession is shown to be about 0.5 mm for the entire vehicle surface at this altitude.

6. DISCUSSIONS

In this work, we first introduce the Park's injection induced turbulence model that describes the eddy viscosity of ablation product gas at the wall surface. We then show the outline of how the model can be implemented into existing CFD code. The results of validation work using existing experimental data obtained in hypersonic wind tunnel are indicated. The eddy viscosity of the injected gas is found to enhance convective heating rate at the wall surface,
and also to promote earlier turbulent transition of the boundary layer. It is shown that the augmented heating rate due to turbulence effect is well reproduced at the stagnation point, but slightly overestimated in the downstream region.

Next, the calculated results of trajectory-based aerodynamic heating analyses for Pioneer-Venus probe vehicles are shown. It is indicated that higher heating rate appears in the downstream region because of the turbulence effect due to ablation product gas. A good agreement of temperature history at the embedded thermocouple positions for the North Probe is obtained. Finally, a trajectory-based analysis of heating environment for MUSES-C entry capsule is attempted. Again, we obtain the elevated temperature both at the stagnation point and in the downstream region due to turbulence effect. It is suggested that the amount of turbulent heating exerted at the stagnation point is similar to that from the catalytic recombination reactions of O and N atoms at the wall. The overall amount of surface recession is found to be almost constant and is about 0.5 mm at the altitude of 49 km.

It should be noted that the cause of the observed higher heating rate in the downstream region both in the Pioneer-Venus mission and also in the Galileo mission is not yet identified as the effect of earlier transition and enhancement of turbulence intensity due to ablation product gas. The effect of radiation and spallation should be studied in detail. However, the results obtained so far strongly suggest that the effect of turbulence due to ablation product gas is rather dominant and at least cannot be neglected.

The results shown here are part of our continuous efforts to develop an analysis code for designing ablative heatshield for a planetary entry vehicle. For this purpose to be accomplished, we obviously need more relevant experimental/flight data that can be used in validation study. The MUSES-C sample return mission is certainly the one that can provide the detailed flight data needed in such work. We also expect to have a similar data, though limited, from the entry flight of Stardust sample return capsule that is scheduled to enter into Earth atmosphere in year 2006. We have already attempted a trajectory-based analysis of heating environment for Stardust entry capsule (Suzuki T. et al. 2002b), and wait for an opportunity to compare the obtained prediction with actual flight data. It is certainly interesting to see whether higher heating rate really appears at the downstream frustum region in these sample return capsules.

ACKNOWLEDGMENT

The author would like to thank many graduate students of the Department of Aeronautics and Space Engineering, Tohoku University, who have contributed in obtaining various results shown in this article.

REFERENCES

An Experimental Study on the Boundary Layer Transition due to Gas Injection from Capsule-shape Body Surface

By

Tetsuya YAMADA*, Hiroyuki OGAWA*, Satoshi NONAKA*, Yoshifumi INATANI*, Kazuyuki NAKAKITA† and Takashi YAMAZAKI†

(1 February 2003)

Abstract: In order to study the onset of the ablation gas-induced boundary layer transition during the reentry phase, heat flux measurement through infrared thermography has been conducted on the capsule-shape body with gas injection from the porous material surface in the shock wind tunnel. In the present simulation experiment, Reynolds number and the ratio of the gas injection mass flow rate to the main mass flow is parametrically changed as similarity law parameters taking account of those in the flight environment; not a few assumptions are applied without verification; the effect of temperature of the boundary layer and the wall etc. At the local Reynolds number of $4 \times 10^4$ and the ratio of gas injection to main flow is about 2%, the heat flux enhancement at the skirt part of the capsule model has been observed and it is considered to be an evidence of the boundary layer transition from laminar to turbulence.

1. INTRODUCTION

The Institute of Space and Astronautical Science (ISAS) has an asteroid sample return mission named MUSES-C, which is planned to be launched in 2003. In the final phase of the MUSES-C, a small capsule with asteroid sample will conduct reentry flight directly from interplanetary transfer orbit (ISAS, 2001). The capsule is exposed to severe aerodynamic heating up to 15 MW/m² due to its high reentry velocity. An ablator heatshield is suitable thermal protection for such reentering vehicles in high-heat flux environment. Actually a carbon phenolic ablator is adopted for the heatshield of the MUSES-C reentry capsule (Yamada, et. al., 2002).

However, it is reported that ablation gas injected from the surface can induce the boundary layer transition, which leads to enhancement of the convective heating rate. From the flight data obtained in the Pioneer-Venus mission, it was observed that the temperature is significantly

* Institute of Space and Astronautical Science, 3-1-1 Yoshinodai, Sagamihara, Kanagawa, JAPAN.
† National Aerospace Laboratory, 7-44-1 Jindaiji Higashimachi, Chofu, Tokyo, JAPAN.
elevated at the downstream frustum region than that at the stagnation region (Wakefield & Pitts, 1980). The heating enhancement observed in the downstream region of could be caused by the boundary layer transition induced by ablation gas injection (Park, 1999).

Several experimental studies on laminar-to-turbulent transitions over a body with surface mass injection have been conducted by means of porous materials. In case of the boundary layer transition, the laminar heat transfer is predicted to be increased up to the turbulent heat transfer. Kaattari (1978), calculating the heat transfer rate from calorimeter temperature versus time transients, showed the early boundary layer transition due to the gas injection. In the present study infrared camera has been used to measure the heat transfer rate based on the time profile of the surface temperature, which is desirable to eliminate the effect of the surface roughness of the calorimeter on the transition.

2. FLIGHT ENVIRONMENT FOR THE REENTRY CAPSULE

On a experiment on the boundary layer transition, it is essential to duplicate the Reynolds number of the flight environment in the windtunnel. The flight environment of the MUSES-C reentry capsule is shown in Fig. 1.

The Reynolds number in the boundary layer is defined based on temperature, pressure, and the velocity at outer edge of the boundary layer with the thickness as a reference length. The boundary layer Reynolds number is estimated as follows;
1) The pressure distribution on the capsule surface is estimated based on the Newtonian-theory.
3) Calculation of the boundary layer Reynolds number based on the Cohen's correlations as for the density and the transport properties of the high temperature air (Cohen, 1960).

The Reynolds number of the boundary layer at a given altitude along the reentry trajectory is shown in Fig. 2. The boundary layer Reynolds number shows higher value at the lower altitude, and the value at the shoulder portion of the capsule-shape is higher than that in the stagnation part. The boundary layer Reynolds number at the time of the maximum heat transfer is estimated to be $1.7 \times 10^4$. 

![Graphs showing heat flux, dynamic pressure, and Reynolds number over time.](image-url)
3. EXPERIMENTAL APPARATUS AND PROCEDURE

3.1 Shock Wind Tunnel

The middle-size shock wind tunnel at NAL (National Aerospace Laboratory) operates in two modes; one is normal shock wind tunnel operation, and the other is QUIC (Quasi Isentropic Compression) mode (Soga, Inoue, et. al., 1992). The special feature of the QUIC mode is as follows; 1) It can operate with long duration up to 40 ms because it does not use double diaphragms. 2) It is quasi-isentropic compression due to relatively weak shock wave properga-
tion. In order to obtain higher Reynolds number, the operation condition with the higher total pressure and the lower total temperature is required. Taking account of the operation envelope of the shock tunnel in QUIC mode, the highest Reynolds number is accomplished with high P4 (pressure at the high pressure storage tank) and high P4 to P1 (pressure at the low pressure tube) as shown in Table 1. It is clear that the shock tunnel in QUIC mode is ideal for the simulation of the Reynolds number of the flight environment of the reentering capsule body.
3.2 High Speed Heat Flux Measurement by Infrared Thermography

In general the boundary layer transition from laminar to turbulence is considered to lead to the enhancement of the heat flux on the surface. We will obtain some clue as for the boundary layer transition by measuring the heat flux distribution. Inoue et. al. established the infrared (IR) thermography technique by means of the infrared camera in the hypersonic wind tunnel (Inoue, Yoshizawa, et. al). The duration of the hypersonic wind tunnel with duration of 30 sec enables the thermography system to obtain 2 dimensional temperature distribution images. The thermography technique needs to be improved so that data acquisition can be completed within several tens of mili-seconds ms within the duration of shock wind tunnel in QUIC.
mode. In the present study, the IR thermography system was expanded so that it can scan 1-dimensional temperature profile in 0.4 ms; The 2-dimensional image must be given up due to the CPU speed of the data handling unit. As shown in Fig. 3, the scanning of the 1 line takes 0.4 ms (Frequency : 2.5 kHz), and 136 lines in the time-progress-wise direction are integrated to 1 frame. The resolution of the temperature measurement is 0.08° C (12 bit data for 100°C). The heat flux is deduced from the time profile of the surface temperature by means of Shultz and Jones Method (Shultz and Jones 1973) expressed as

$$ q = \sqrt{\frac{\rho c k}{\pi}} \left[ T(t) + \frac{1}{2} \int_0^t \frac{T(t) - T(\tau)}{(t-\tau)^{3/2}} d\tau \right] \quad (1) $$

where $\rho$: density, $c$: specific heat, $k$: heat conductivity, $T$: measurement time, and $T(t)$: temperature at time $t$. Here the aerodynamic heating rate is assumed to be constant during the measurement time $\tau$.

The above equation is valid under the assumption of infinite slab approximation that means the thickness of the slab has no effect on the conduction of the heat. The thickness of the material must satisfy the next relation between thermal diffusivity ($\alpha$) and the duration ($t$) of the aerodynamic heating expressed as

$$ thickness > 4\sqrt{\alpha \cdot t} \quad (2) $$

The thermal diffusivity of the material used for the experiment needs to be low enough that the thermal penetration depth of the material due to the aerodynamic heating is small in comparison with the material geometrical thickness.

### 3.3 Test Model with Gas Injection

The ablation gas from the surface of the reentering body is simulated by the gas injection from the surface of the porous material of the test body under the assumption as follows; 1) The air in the room temperature is used for the injection gas. 2) The effect of the injected gas temperature and the wall temperature is negligible.

Because the diameter of the carbon fiber ranges 10 to 20 $\mu m$ in case of the carbon phenolic ablator such as used of the MUSES-C reentry capsule, the gas should be injected from the holes with the diameter of the same order with the fiber. Thus the porous ceramics made of Aluminum Borate ($9Al_2O_3 \cdot 2B_2O_3$) has been used for the test model, of which mechanical and thermal properties are summarized in Table 2 (Iizuka, 1998).

<table>
<thead>
<tr>
<th>Chemical Composition</th>
<th>Aluminum Borate ($9Al_2O_3 \cdot 2B_2O_3$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Porosity</td>
<td>72% (normally 80-20%)</td>
</tr>
<tr>
<td>Allowable Bending Stress</td>
<td>10-200 [MPa]</td>
</tr>
<tr>
<td>Heat Conductivity ($V_f = 25%$)</td>
<td>(0.528 W/mK (600 deg C)</td>
</tr>
<tr>
<td>Thermal Expansion Coeff ($V_f = 25%$)</td>
<td>4.8E-6 [/deg C]</td>
</tr>
<tr>
<td>Maximum Temperature</td>
<td>1300 [deg C]</td>
</tr>
</tbody>
</table>
A cross sectional view of the test model used for the present study is shown in Fig. 4. Because the total gas injection mass flow needs to be within the gas supply ability and the evacuation ability of the wind tunnel, the porous material is installed only in the strip region on the surface with the width of 7 mm.

As a similarity law, the ratio of the gas injection mass flow rate to the main mass flow (\( \rho u \): density multiplied by flow velocity) is parametrically changed taking account of those in the flight environment.

Figure 5 shows the experimental apparatus arrangement in the shock wind tunnel facility. The mass flow rate of the injection gas is estimated based on the pressure decay profile of the gas-supply mixing chamber installed upstream of the test model.

**Gas Injection**

The mass flow rate of the gas injection was correlated to the model surface pressure (Ps) and the model chamber pressure (Pc) as follows;

\[
\dot{n} = C \left( P_{chamber}^2 - P_{surface}^2 \right) \tag{3}
\]

The coefficient C had been measured in advance to the experiment. As shown in the shock tunnel operation condition, the surface impact pressure on the model is about 10 kPa while the model chamber pressure ranges 100 to 500 kPa. Because the mass flow rate is proportional to the difference of their square, the gas injection rate distribution due to the impact pressure distribution is trivial at most 1%. Figure 6 shows the mass flow density vs. the model chamber pressure characteristics.
4. EXPERIMENTAL RESULT AND DISCUSSION

4.1 Experiment Sequence

Figure 7 shows a typical time profile of the wind tunnel operation together with the pressure of gas-supply mixing chamber. After appropriate time delay of the valve-open signal for the high-pressure tank (P4), the solenoid-valve downstream of the injection chamber is opened and the gas begins to blow out from the test model. Within about 100 ms, the main flow is build up as recognized from the pressure at the test section. Fig. 8 show a typical Schlieren photograph of the test piece without gas injection. Any differences in the Schlieren photos with / without gas injection are not clearly observed, perhaps because the density gradient is too small to be detected in the shock wind tunnel.

4.2 Blockin Effect of Stagnation Point Heat Transfer

The effect of the mass addition on the stagnation point heating (Marvin, 1967) is generally well predicted for laminar flow by

\[ \psi = \frac{\dot{q}_{\text{stagnation}}}{\dot{q}_{0,\text{stagnation}}} = 1 - 0.72B + 0.13B^2 \]

\[ B = \frac{\dot{m}\Delta H}{\dot{q}_{0,\text{stagnation}}} \]

where \( \dot{m} \) is mass addition rate and \( \Delta H \) is the heat transfer driving potential defined as the difference between the wall enthalpy and the stagnation enthalpy. The term \( \psi \) is the ratio of the heat transfer \( \dot{q}_{\text{stagnation}} \) at the presence of mass addition with respect to the heat transfer
Experimental Study on Boundary Layer Transition

Fig. 9: Correlation of Convective Heating Rate Ratio $\psi$ with Mass Addition Parameter B.

$q_{0,\text{stagnation}}$ with no mass addition. The independent variable $B$ is the blowing parameter. $\psi$ of all data have been plotted versus $B$ together with Marvin's correlation as shown in Fig. 9. In the present experiment conducted in the shock windtunnel, the blowing parameter ranges 0 to 2.5 and then $\psi$ the ratio of the heat transfer ranging 1 to 0.1 is well correlated to Marvin's correlation though not a small dispersions are seen in high B region.

4.3 Heat Flux Distribution and the Effect of the Gas Injection

The surface heat flux distributions normalized by the stagnation value are shown in Fig. 10 to Fig. 12 through with the parameter of the ratio of injection mass flow rate ($m$) to the main mass flow rate ($M$).

When the gas injection rate equals to zero, the heat flux distribution is almost identical to Lee's distribution. The qualitative features in case of gas injection are as follows; 1) The heat flux distributions are uniformly decreased with increased $m/M$ ratio regardless of the Reynolds number. 2) The heat flux distributions in the high and middle Reynolds number exhibit unique aspect that the heat flux downstream of the model juncture point (frustum part) onsets increase beyond a certain $m/M$ ratio. For example, the heat flux in the high-Reynolds condition start to increase at and beyond $m/M$ is 1.81%. Moreover the heat flux at the frustum part is higher than the value at the low $m/M$ ratio such as 1.1% and 0.65%. Actually it is observed that the heat flux is below zero in a some portions, this is considered to be caused by the cold gas injection.

This phenomenon that the heat transfer increase at the frustum region is considered to be the heat flux enhancement and it might be reasonable that this is an evidence of the boundary layer transition due to ablation-simulated gas injection.
4.4 Boundary Layer Transition along the Reentry Trajectory

On the basis of the criterion described above, the onsets of the boundary layer transition are mapped along the reentry trajectory of the MUSES-C reentry capsule as shown in Fig. 13. The vertical axis denotes the \( m/M \) ratio (ablation mass flow rate to main mass flow) based on the
Fig. 12: Heat Flux Distribution along the Surface (High Re).

Fig. 13: Onset Mapping of the Boundary Layer Transition along the MUSES-C Reentry Trajectory (MC-005MJ).
ablutor analysis along the reentry trajectory. Judging from this figure, we might conclude that the possibility of the onset of the boundary layer transition and resulting heat flux enhancement is small during the reentry.

5. CONCLUSION

In order to study the onset of the ablation gas-induced boundary layer transition during the reentry phase, heat flux measurement has been conducted by means of gas injection from the test model made of porous material in the shock wind tunnel through infrared thermography. In the present simulation experiment, Reynolds number and the ratio of the gas injection mass flow rate to the main mass flow is parametrically changed as only similarity law parameters taking account of those in the flight environment; not a few assumptions are applied without verification; the effect of temperature of the boundary layer and the wall etc. Although authors recognizing these insufficiency would like to conclude as follows; At the local Reynolds number of \(4 \times 10^4\) and the ratio of gas injection to main flow is about 2%, the heat flux enhancement at the frustum part of the capsule model has been observed and it is considered to be an evidence of the boundary layer transition form laminar to turbulence.

REFERENCES


