THE SIMULATION OF TRANSIENT TEMPERATURE DISTRIBUTION INSIDE A BLUNT BODY NOSE AND AN INTERNAL DEVICE USING A FAST PROPOSED ENGINEERING METHOD IN SUPERSONIC AND HYPERSONIC FLOW REGIMES

Amir Yousef Sajjadi and Mostafa Hoseinalipour

Mechanical Engineering Department, Iran University of Science & Technology
Tehran, Iran
E-mail: aysajjadi@iust.ac.ir

Abstract

The modeling of Aerodynamic heating of supersonic and hypersonic flights has been under intensive consideration in recent years. In present work an Engineering fast code is developed for calculating transient temperature distribution inside blunt nose bodies and internal devices in high Mach flows, considering aerodynamic heating effects. The code is evaluated using some existing experimental and numerical data in the literature for existing first stage of Pegasus stage booster. Then the code is used for prediction of temperature variation of two other configuration and the results are presented. In proposed method the flow parameters are computed using inviscid-boundary layer analysis, and then the energy equation inside the skin and internal device is computed numerically. The flow field around the blunt body is divided into three rejoiners, namely, stagnation, near stagnation and body region. The local slope method is used for inviscid flow and approximation methods applied for boundary layer near the surface and appropriate correlations are used to predict convention heat transfer coefficient in various regions around an axisymmetric body moving at some moderate and high mach numbers, at mid-altitude without ablation and sleep and free molecule. The temperature variation inside the body skin is also calculated by solving the Fourier’s heat conduction equation. For internal element two-dimensional energy equation in cylindrical coordinates is used which is coupled with Fourier’s equation. For this reason, in the present study, a finite difference method as a numerical tool is developed to solve the energy equation in an axisymmetric body with a given corresponding boundary conditions. Thermo physical properties of the air are assumed to vary with flight height and temperature. The new approach in this investigation is computing radiosity factors for different shapes and calculating temperature distribution in the internal elements of a high speed-flying object.

Key words: Aerodynamic heating, Shape factor, Hypersonic speed

1. INTRODUCTION

The thermal design of hypersonic vehicles involves accurately and reliably predicting the convective heating over the surface of the vehicle. Such results may be obtained by numerically solving the Navier-Stokes (NS) equations [1] or one of their various subsets such as the parabolized Navier-Stokes (PNS) [2] and viscous shock layer (VSL) equations [3, 4, 5] for the flow field surrounding the vehicle. However, due to the excessive computer run times and storage requirements of these CFD (computational fluid dynamics) approaches, they are impractical for the preliminary design environment where a range of geometries and flow parameters are to be studied. Even coupled solutions of the outer inviscid region described by the Euler equations and the inner viscous layer described by the boundary layer equations may be too computer intensive for preliminary design [6, 7].

On the other hand, engineering inviscid-boundary layer methods [8, 9, 10, 11, 12 and 13] have been demonstrated to adequately predict the heating over a wide range of geometries and aero thermal environments. Various approximations in the inviscid and boundary layer regions reduce the computer time needed to generate a solution. This reduction in computer time makes the engineering aero thermal methods ideal for parametric studies [14].
In this paper the results of an Engineering method for simulation of Aerodynamic heating over conical and blunt nose bodies are presented. In proposed method the flow around the body is solved by Fast Engineering approximation. The inviscid flow is calculated using local slope methods [15] and the boundary layer is solved approximately for plate and applied for this surface by correction factor. For temperature variation inside the surface 1D numerical finite difference method is used, and also for internal device the axisymmetric finite difference method is applied. The radiation exchange between the outer surface and atmosphere, also between inner surface and the board is considered in the proposed formulation.

To predict the temperature distribution inside the nose, radiosity factors were computed using ANSYS software and then shape factors were determined [16].

The geometry is presented in figure (3). The nose is sphere and the internal device is considered cylindrical.

2. INVISCID ANALYSIS

Local slope methods are approximate methods to compute pressure coefficients on a given segment of a body or wing surface based on a free stream mach number and the local slope of a segment of the body or wing with respect to the free stream direction. Several of these methods are dependent on a wedge or cone solution of the inviscid equations of motion. Local slope methods in general are fairly simple and straightforward yet yield reasonable results for force coefficients in many cases.

For bodies with blunt nose or truncated leading edges or tips, modified Newtonian theory (MNT) has proved to be a powerful, easy to use, and an accurate complement to perturbation methods and local slope methods.

Real gas effects is very important in Aerothermodynamics analysis. In present code real gas effects are considered using curve fits from [17] for Mach numbers more than 5.

3. VISCOUS ANALYSIS

An approximate convective-heating method is used to predict transient surface temperature and heating rates for two and three-dimensional stagnation points, both for laminar and turbulent flows. As the viscous region is assumed to be thin in comparison with the shock layer, the appropriate governing equations are the 3-D boundary layer equations [18]. Properties at the boundary layer edge are obtained using the results of inviscid solution. An axisymmetric boundary layer method is applied along an inviscid surface streamline. Although the axisymmetric boundary layer equations may be integrated numerically, a set of approximate convective heating equations developed by Zoby [19] provides accurate surface heating rates with a minimum amount of computational effort. These equations are used here since they are consistent with the nature of the used inviscid method.

3.1. STAGNATION POINT

The equation used to calculate surface temperatures and heat flux for three-dimensional stagnation points is [20]:

\[ q = F \cdot H \cdot (h_1, h_t) - \beta T_w^4 + S \]  

and for two-dimensional stagnation points is:

\[ q = F \cdot H \cdot (h_1, h_t) - \beta T_w^4 + S \]  

S in equation 1 and 2 stands for solar and nocturnal radiation input if required. This term is negligible except for low-speed flows. The term \( \beta T_w^4 \) is the heat lost by radiation from the surface of the body to the atmosphere.

In this model, the heat transfer coefficients are calculated using the method proposed by Fay and Riddell [21] for three dimensional stagnation points. The method of Beckwith [22] is used for two-dimensional stagnation points. The equation given by Fay and Riddell for a Lewis number of 1.0 (no dissociation) and a Prandtl number of 0.71 is as the following:

\[ h = 94(\rho_o \mu_o)^{1/2}(\rho_1 \mu_1)^{1/2} \left( \frac{du}{dx} \right)_{x=0} \]  

and the equation given by Beckwith for a Lewis number of 1.0 and a Prandtl number of 0.71 may be written as

\[ h = 94(\rho_o \mu_o)^{1/2}(\rho_1 \mu_1)^{1/2} \left( \frac{du}{dx} \right)_{x=0} \]  

The velocity gradient is given by

\[ \left( \frac{du}{dx} \right)_{x=0} = \frac{1}{R} \left( \frac{2L p_1 - p_v \gamma}{\rho_o} \right) \]  

and the stagnation enthalpy \( H_{st} \) for both two and three-dimensional flow is calculated using the following equation:

\[ H_{st} = H_s + \frac{v^2}{2g} \]
where the subscript “2” denotes the conditions calculated behind the normal shock. The wall enthalpy $H_w$ is given by $H_w = f(T_w, P_i)$ and is determined from real gas tables presented in [17].

### 3.2. TRANSIENT HEATING EQUATION

The following equation is used to calculate transient surface temperatures and heat flux:

$$q = F*H*(h_w - h_{w,2}) - \beta T_w^4 + S$$  \hspace{1cm} (7)

For laminar heat transfer:

$$h = (F)0.332 \sqrt{\frac{\rho' \nu V}{X}} (Pr_w)^{-0.6}$$

Equation 7 is based on the Blasius incompressible skin friction formula [24] and is related to heat transfer by a modified Reynolds analogy by the following formula:

$$ST = RA \frac{C_f}{2}$$  \hspace{1cm} (8)

where $(Pr_w)^{-0.6}$ is the modified Reynolds analogy factor and the Stanton number “ST” is given by:

$$ST = \frac{h}{\rho V}$$  \hspace{1cm} (9)

And the Blasius skin friction formula is:

$$\frac{C_f}{2} = 0.332 (Re_e)^{-1/2}$$  \hspace{1cm} (10)

Compressibility effects are accounted for by Eckert’s reference enthalpy method [25, 26], and the flow properties are evaluated at the reference enthalpy given by the following equation:

$$H^* = 0.5(H_w + H_L) + 0.22(H_R + H_L)$$  \hspace{1cm} (11)

where

$$H_R = H_L + \sqrt{Pr_{w,2}} \frac{V_L}{2gJ}$$

and the values of, $H_w, H_L$ and $T^*$ are obtained from real gas tables, [17].

The turbulent heat transfer coefficient is obtained by solving for the turbulent skin friction coefficient and then relating the skin friction to heat transfer by a modified Reynolds analogy [27]. The heat transfer coefficient calculated by the van Driest method is then given by the following equation:

$$h = F \frac{C_f \rho V}{2(Pr_{w,2})^{1/2}}$$  \hspace{1cm} (13)

### 4. INTERNAL BOARD AND INSIDE BODY EQUATIONS

A one dimensional transient energy equation is solved in order to calculate the time evolution of temperature inside the nose body and also inside the internal device.

The boundary conditions for the external surface are radiation to atmosphere and aerodynamic heating into the surface and the boundary conditions for internal surface is Radiation to internal device. The boundary condition for the internal device equations is radiation to the internal surface.

To determine the radiative energy exchange between internal surfaces, the radiation shape factor that is a geometric parameter, can be obtained by using the equation below [16].

$$F_{ij} = \frac{1}{A_i A_j} \int \frac{\cos \theta_i \cos \theta_j}{\pi R^2} dA_i dA_j$$  \hspace{1cm} (16)

### 5. COMPUTATIONAL PROCEDURE

The governed equations 14, and 15 are coupled with equations 1 to 13 and are solved explicitly using finite difference method. The view factors in eq16. are obtained by ANSYS software using Network Analysis methods [28].
6. RESULTS

Figure 1(a, b) represent the Pegasus flight trajectory parameters (i.e.: flight Mach number and altitude versus time) which is used to validate the results for a blunt body which is shown in fig. 2. The variation of temperature for stagnation point and point c on the surface of the body, are compared with experimental data in ref. [29], and are shown in fig 3 and 4, representing a good consistency. Then the method was applied to the same configuration with an assumed electronic device inside the nose body. To validate the view factors for temperature distribution in cylinder inside the missile, view factors for a simple case were computed by ANSYS and compared with ref [16]. Estimated error was about 4 percent which is acceptable. The independency of grid points where examined using 4 and 8-point grid in the skin and 432 and 520 grid points for internal board. Temperature distribution inside skin of missile is represented in Fig 5. Two different nose shapes were examined to see the nose bluntness effects. The first case is a blunt nose \( \frac{R_n}{R_b} = \gamma \) in which \( R_n \) is nose radius and \( R_b \) is body radius. The second case is a cone with sharp leading edge \( \frac{R_n}{R_b} = 0.05 \) (Figure 6).

The temperature distribution inside the internal electronic device is presented in figures 7 and 8 for sharp and blunt nose, respectively. As it is expected the maximum temperature inside the electronic device for sharp nose is greater than the blunt ones.
The same conclusion can be deduced from figure 9 and 10 that present the stagnation rejoin temperature of two different noses.

Conclusion

A code was developed to predict the transient temperature distribution through the surface of missile and its internal devices. Inviscid flow is calculated using local slope inclination methods, which can give acceptable, results for free a stream property. The main point of this modeling is to predict the temperature distribution in surface and internal board of missile during its trajectory. This model can predict blunt nose missiles as well as sharp one.

REFERENCES


Fig. 1- Flight trajectory parameter. A) MACH, TIME  
B) ALTITUDE, TIME

Fig. 2- blunt body and point c along it

Fig. 3- Comparison of stagnation point temperature

Fig. 4- Comparison of temperature for point c along missile.

Fig. 5- Temperatures distribution inside the skin

Fig. 6- spherical and sharp edge missile

Fig. 7- Temperature distribution in internal board of sharp nose missile

Fig. 8- Temperature distribution in internal board of spherical nose missile

Figure. 9- stagnation region temperature variation for sharp nose missile

Figure. 10- stagnation region temperature variation for blunt nose missile