CALCULATING METHOD OF AERODYNAMIC HEATING FOR HYPERSONIC AIRCRAFTS

Ji Weidong(季卫栋), *Wang Jiangfeng*(王江峰)

(College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing, 210016, P. R. China)

Abstract: A new calculating method of aerodynamic heating for unsteady hypersonic aircrafts with complex configuration is presented. This method, which considers the effects of high temperature chemical non-equilibrium and the heat transfer process in thermal protection structure, is based on the combination of the inviscid outerflow solution and the engineering method, where the Euler solver provides the flow parameters on boundary layer edge for engineering method in aerodynamic heating calculation. A high efficient interpolation technique, which can be applied to the fast computation of longtime aerodynamic heating for hypersonic aircraft, is developed for flying trajectory. In this paper, three hypersonic test cases are calculated, and the heat flux and temperature distribution of thermo-protection system are shown. The numerical results show the high efficiency of the developed method and the validation of thermal characteristics analysis on hypersonic aerodynamic heating.

Key words: hypersonic aircraft; aerodynamic heating; fluid-structure coupled analysis; chemical non-equilibrium effects; coupling of numerical and engineering methods

CLC number: V211.3 **Document code**: A **Article ID**: 1005-1120(2013)03-0237-06

INTRODUCTION

With the development of the techniques of hypersonic vehicles, aero-heating and thermal protection are not just for re-entry vehicles, but for other types of aircrafts, such as those flying in atmosphere in higher supersonic or hypersonic speed. To develop a high efficiency method of aero-heating analysis for thermal protection system is very important in hypersonic vehicle design engineering[1].

The methods for the thermal environment analysis are numerical simulation (computational fluid dynamic (CFD)), engineering approximation[2], ground wind tunnel experiments and free-flying experiments[3]. The last two methods are too costly to suit the early engineering design and modification.

CFD method is well-known as solving the Navier-Stokes equations and for its various simplified form. The CFD methods have high precision, but the computation efficiency is terrible, especially in unsteady flying status [4].

Because of its simple calculation process and high computational efficiency, the engineering approach has been developing rapidly since it was put forward. The aerodynamic heating problems are divided into two categories according to existence of similar solutions of the boundary layer equations[5]. For the flat-panel flow, Eckert, Spalding-Chi, Van Driest and Zoby[6-7] made a lot of research to give out the laminar and turbulent heat flux engineering formula. A fast and effective method for dissimilarity solution is put forward by Lees, Van Driest, Lee and Faget[8]. There are axisymmetric simulation methods, equivalent cone methods and experimental data related methods for the aerodynamic heating calculation while the hypersonic aircraft has angle of attack. Engineering approach still has its limitations, such as the computation of boundary layer outer edge parameters, the partition of the air-

Received date: 2013-04-20; **revision received date**: 2013-07-08

Corresponding author: Wang Jiangfeng, Professor, E-mail: wangjf@nuaa.edu.cn.

craft with complex shape (cracks, bumps, irregular shapes, etc.), the correction of the formulas and the results based on the engineering experiences.

In order to find a method coupling the respective advantages of CFD method and engineering approach, the method used in this paper is based on the Prandtl boundary layer theory. The entire flow field is divided into two parts: Inviscid flow out of the boundary layer and viscous flow within the boundary layer. As a result, the process of solving aerodynamic heating can be simplified as two single steps. One is the solution of the hypersonic inviscid outflow. The other is the computation of the viscous dominated areas within the boundary layer. On the basis of a reasonable approximation, numerical solution in the inviscid outflow and engineering approach in the boundary layer are coupled to compute the high-speed aircraft surface heat flux distribution. And heat transfer equation is considered in order to analyze the thermal structure performance [9].

The characteristic of the method in this paper is to combine the respective advantages of the numerical simulation with the pure engineering approach. It makes up for the defects of numerical simulation methods which have low efficiency, long life cycle and high cost, and expands the scope of application of the pure engineering approach. The method can be easily used to compute the aerodynamic force and aerodynamic heating of complex geometries in complex flow.

1 AERODYNAMIC HEATING CALCULATION PROCESS

Aerodynamic heating calculation method used in this paper is based on a new method, which couples the CFD method with the engineering approach. The entire flow field is divided into two parts based on the Prandtl boundary layer theory. The thickness of the boundary layer is ignored because it is very thin. Namely the shape of the outer edge of the boundary layer is the same with the shape of aircraft. The parameters out of the boundary layer, such as pressure, temperature and velocity, are computed by CFD methods. The heat transfer characteristics within the boundary layer and the structure of thermal protection system are worked out by engineering approach. These two methods are coupled alternately forward.

2 INVISCID OUTFLOW

To solve the invisid outflow is to resolve the Euler equations of fluid dynamics in the flow field. The CFD methods of Euler equation are quiet mature and perfect. The solution of the inviscid out flow in this paper is based on the hybrid grid and parallel computing. The space discretization uses AUSM$^+$ scheme [10] and the time discretization uses fourth-order Runge-Kutta method. The states of inviscid outflow solutions are determined by the type of aerodynamic heating such as transient heating, cruise state or trajectory state, etc. The flow parameters of grid points on body surface are taken as the interpolation parameters required by engineering approach in the boundary layer outer edge. If needed, the interpolation database can be established.

3 ENGINEERING COMPUTING TECHNOLOGY

3.1 Heat transfer in boundary layer

The surface of the hypersonic aircrafts is divided into two parts: The stagnation points and non-stationary zones. For the former, the Fay-Riddell formula is fit for axisymmetric body, of which $Pr=0.71$, $Le=1.0$. The formula[11] is as follows

$$q_{\mathrm{ws}}=0.763\times Pr^{-0.6}\times\left(\frac{\rho_{\mathrm{w}}\mu_{\mathrm{w}}}{\rho_{\mathrm{s}}\mu_{\mathrm{s}}}\right)^{0.1}\times\sqrt{\rho_{\mathrm{s}}\mu_{\mathrm{s}}\left(\frac{\mathrm{d}u_{\mathrm{e}}}{\mathrm{d}x}\right)_{\mathrm{s}}}\times\left[1+(Le^{0.52}-1)\frac{h_{\mathrm{D}}}{h_{\mathrm{s}}}\right](h_{\mathrm{s}}-h_{\mathrm{w}})\tag{1}$$

where ρ_{w} is the density on the object surface, μ_{w} the viscosity coefficient on the object surface, h_{w} the enthalpy on the object surface, ρ_{s} the density in stagnation point, μ_{s} the viscosity coefficient in stagnation point, $\left(\frac{\mathrm{d}u_{\mathrm{e}}}{\mathrm{d}x}\right)_{\mathrm{s}}$ the velocity gradient in stagnation point, and h_D the average air dissociation enthalpy.

For the non-stagnation zone, the engineering corresponding empirical formulas are also given. For the case of zero angle of attack (AOA), there are Lees formula and reference enthalpy method. In this paper, the flat heat transfer model is used to calculate the heat transfer coefficient of the boundary layer, that is

$$a = \frac{1}{2} Pr^{-\frac{2}{3}} c_p^* \ V_c c_{tc}^* \ \rho_c{}^* \tag{2}$$

where a is the heat transfer coefficient, c_p^* the specific heat capacity at constant pressure, V_c the velocity in the outer edge of the boundary layer, $\rho_c{}^*$ the density in the outer edge of the boundary layer, and c_{tc}^* related to turbulent viscosity coefficient.

3.2 Heat transfer in structure

Continuous aerodynamic heating process of hypersonic aircrafts is an unsteady process. The heat flux from external airflow to the aircraft surface is related to the wall temperature. Newton gives out the law of cooling, which states that

$$q_W = a(T_{aW} - T_W) \tag{3}$$

where q_W is the heat passed from the air to the structure, T_W the temperature on the aircraft surface, and T_{aW} the recovery temperature.

Engineering methods are used to solve the flow field aerodynamic heating. At the same time, heat transfer is computed in the thermal protection structure. This process is coupled. On one hand, as the heat flux increases, the wall temperature rises. On the other hand, the increase of the wall temperature has an influence on the heat flux. To compute the surface heat flux, it needs to know the wall temperature. And the latter one also depends on the former. That is the coupling of the heating process.

The specific computing process is:

(1) Give the wall temperature at initial moment and then calculate the heat flux by engineering approach. Take this heat flux as the thermal boundary condition to work out the wall temperature in the next time. Repeat the process and promote the solving by time step.

(2) Within each time step, firstly, take the wall temperature in previous time as thermal boundary condition to get the wall heat flux distribution. Then analyze the heat transfer in the thermal protection structure by considering the heat flux distribution as the thermal boundary condition. Thus the new wall temperature distribution can be used as the thermal boundary condition in solving the flow field in the next time step.

4 HIGH-TEMPERATURE CHEMICAL NON-EQUILIBRIUM EFFECTS

For the freezing catalytic wall, the non-catalytic wall and the limited catalytic wall, the stagnation point heat flux formula given by Fay-reddell has been widely used. But it should be noted that those results are obtained under the two-component assumption (i. e., gas atoms and molecules). That means the oxygen atoms, nitrogen atoms, oxygen molecules and nitrogen molecules are not distinguished when dealing with boundary layer equations. Some papers take the wall catalytic rate of the oxygen atoms and nitrogen atoms into consideration. The ratio of the stagnation point heat flux between the finite catalytic wall with freezing boundary layer and equilibrium conditions are given as[9]

$$\frac{q_{Ne}}{q_{eq}} = \frac{\left[1 + (Le\varphi_0 - 1)\dfrac{h_O^0 \, C_{O,s}}{h_s - h_w} + (Le\varphi_0 - 1)\dfrac{h_N^0 \, C_{N,s}}{h_s - h_w}\right]^{0.5}}{\left[1 + (Le - 1)\dfrac{h_D}{h_s - h_w}\right]^{0.5}} \tag{4}$$

$$\varphi_i = \frac{1}{1 + \dfrac{0.664 Sc^{-2/3}\sqrt{\rho_\omega \mu_\omega \left(\dfrac{du_e}{dx}\right)_s}}{\rho_\omega k_{\omega i}}} \quad i = O, N \tag{5}$$

where q_{Ne} is the heat flux considering the effect of chemical non-equilibrium, q_{eq} the heat flux while ignoring the effect of chemical non-equilibrium, h_O^0 the formation enthalpy of oxygen atoms, $C_{O,s}$ the mass concentration of oxygen atoms, h_N^0 the formation enthalpy of nitrogen atoms, $C_{N,s}$ the mass concentration of nitrogen atoms, and $k_{\omega i}$ the wall catalytic rate.

What we need to determine is the mass concentration of the oxygen atoms and nitrogen atoms at the stagnation point. Equilibrium stagnation point heat flux can be given out according to the Fay-Reddell heat flux formula. In the heat flux ratio relationship, heat flux ratio between Non-balance and balance flow in the course can be determined by the mass concentration of oxygen atoms and nitrogen atoms in stagnation points. Thus heat flux in the non-equilibrium effects can be obtained from the known heat flux in the equilibrium conditions.

5 EXAMPLES AND ANALYSIS

5.1 Case 1: Heat flux

RAMC-II test module is a ball cone. The head radius is 0.152 4 m, the semi-cone angle is 9°, and the total length is 1.295 m. Flight status: Mach number is 4.0, AOA is 0°, flight altitude is 60 000 m. Compared with the result in other paper[12], the tendency matches well. The maximum heat flux calculated in this paper is 14 768 W/m^2. The maximum heat flux is 23 417 W/m^2 according to Lees′s formula and it is 14 227 W/m^2 according Kemp-Riddel′s law[9]. The value in this paper is similar to that of Kemp-Riddel. Fig. 1 is the heat flux distribution of the RAMC-Ⅱ test module. Fig. 2 shows the heat flux distribution in symmetry plane.

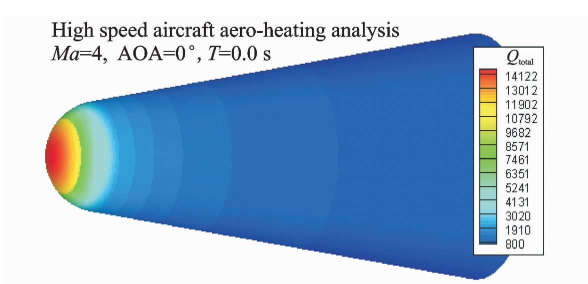

Fig. 1 Heat flux distribution

5.2 Case 2: Long time heat transfer in state of cruise

Case 2 is a winged missile body of complex shape. Fig. 3 shows the configuration of the Case

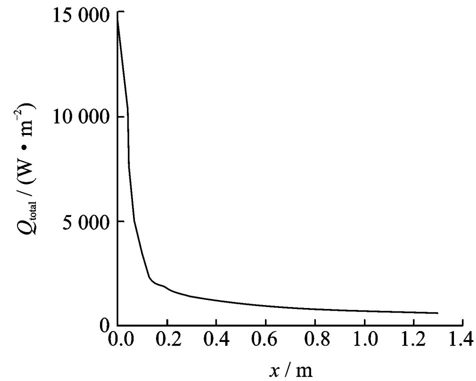

Fig. 2 Heat flux distribution in symmetry plane

2. Flight status: Mach number is 6.0, AOA is 6°, flight altitude is 30 000 m, and flying time is 1 000 s. The entire aircraft is set as one thermal protection system (TPS) section. The materials are divided into 21 layers in the thickness direction. The first layer is aluminum of 0.002 m. The remaining 20 layers are 0.001 m Silica in each layer.

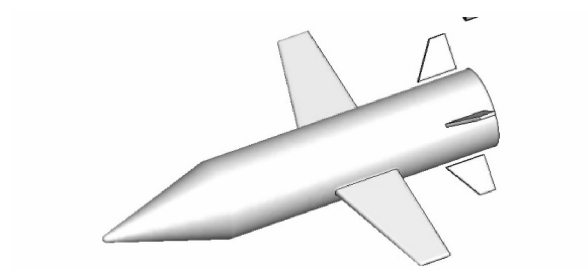

Fig. 3 Outer surface

Case 2 is long time heat transfer of the cruise. The total CPU time is only about 20 min (excluding the CPU time when solving the inviscid outflow). It is much less than the used numerical method (solving the NS equations and structural equations). Fig. 4 is the temperature distribution of outer surface in 1 000 s. It can be seen that the surface temperature in the head is about 950 K and the maximum temperature is nearly 1 250 K. Fig. 5 is the temperature distribution of inner surface in 1 000 s. The maximum temperature of inner surface is nearly 1 050 K. Clearly, TPS reduces the temperature significantly. This case demonstrates that the method is effective for long aerodynamic heating of high-speed aircraft with complex shape. It shows that this

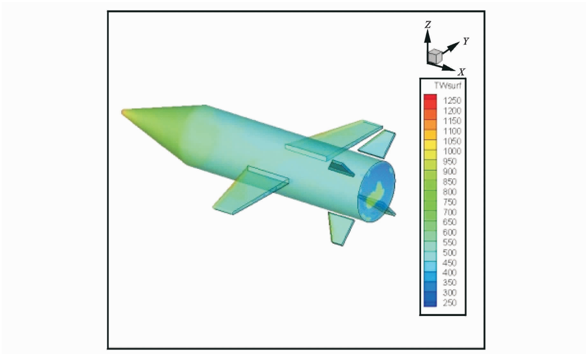

Fig. 4 Temperature distribution of outer surface in 1 000 s

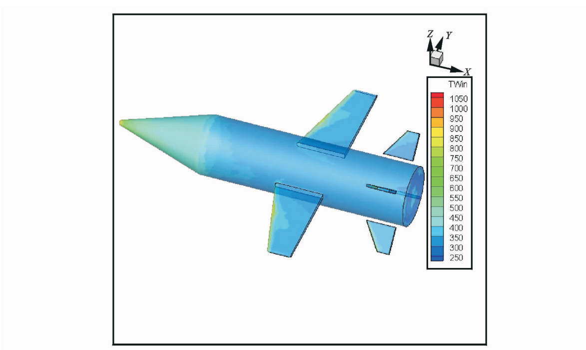

Fig. 5 Temperature distribution of inner surface in 1 000 s

method is fast and efficient, and meets the engineering requirements.

5.3 Case 3: Chemical non-equilibrium effect

The shape of Case 3, RAMC-II test module, is a ball cone. The head radius is 0.152 4 m. The semi-cone angle is of 9°. The total length is 1.295 m. Flight status: Mach number is 5.0, AOA is 6°, flight altitude is 60 000 m, and flying time is 1 000 s.

Case 3 demonstrates the impact of chemical non-equilibrium effects on aerodynamic heat transfer. In this case, only the oxygen atoms and nitrogen atoms are taken into consideration. Fig. 6 is the variation of molar ratio with temperature. Fig. 7 is the changes in mass concentration of oxygen atoms and nitrogen atoms with temperature. Figs. 8—10 are the comparisons of heat flux and surface temperature according to whether to consider the effect of chemical non-equilibrium. It can be seen that the effect of chemical non-equilibrium allows heat flux decreased and the surface temperature drops. In Fig. 9, the maximum surface temperature of the aircraft is 839.055 K. And it is 821.679 K in Fig. 10. Due to the low Mach number and high altitude, the chemical reaction is not obvious. This leads to a low concentration of nitrogen and oxygen atoms and a small temperature difference.

These three cases show that coupling inviscid numerical solution with aerodynamic heating engineering approach forms a quick computing tech-

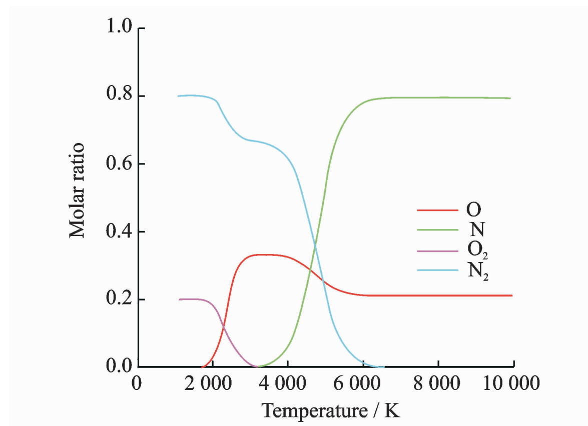

Fig. 6 Molar ratio

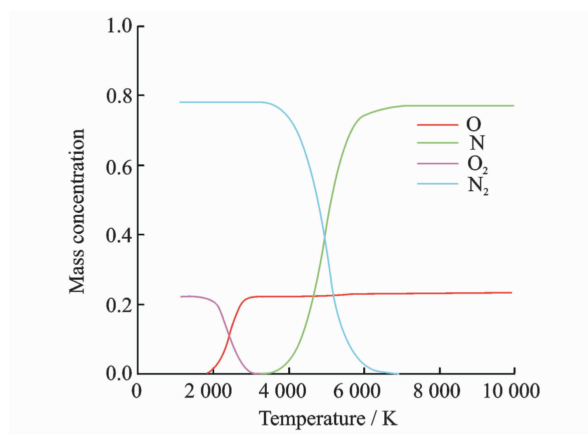

Fig. 7 Mass concentration

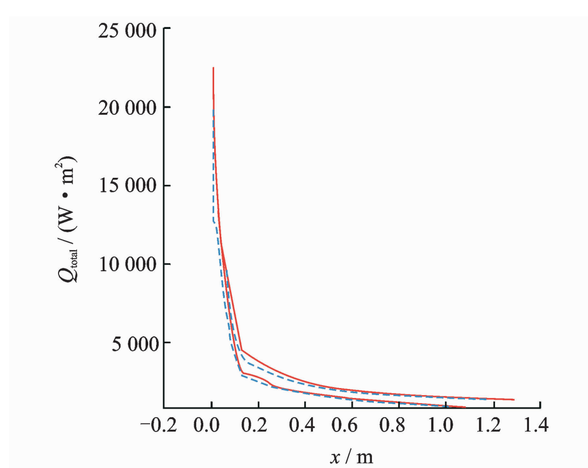

Fig. 8 Heat flux

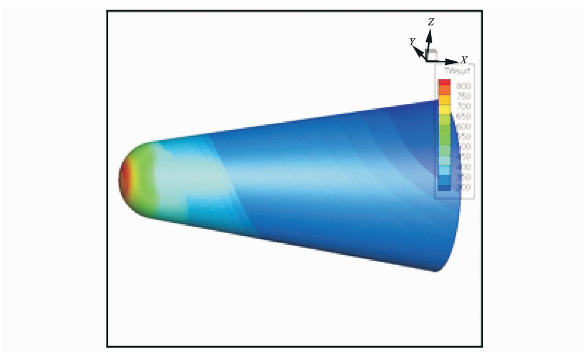

Fig. 9 Temperature distribution in 1 000 s

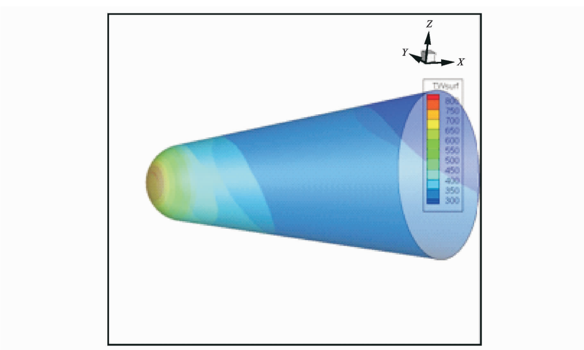

Fig. 10 Temperature distribution in 1 000 s (Considering chemical non-equilibrium effects)

nology for the surface heat flux, the aerodynamic heating and the aerodynamic force. The method is fast and efficient, and meets the engineering requirements well.

6 CONCLUSION

The main computing technology used in this paper is based on the Prandtl boundary layer theory. The entire flow field is divided into two parts: Inviscid flow out of the boundary layer and viscous flow within the boundary layer. Thus the problem can be simplified as two small questions. One is to solve the inviscid flow out of the boundary layer. The other is solving the viscous dominant region within the boundary layer. The inviscid numerical solution, the aerodynamic heating engineering approach and the heat transfer in the thermal protection structure are combined to fast compute the long aerodynamic heating for hypersonic vehicle. The accuracy of the method developed in this paper well meets the engineering requirements, and the computational efficiency is much higher than that of the numerical method which is to solve the N-S equations. Currently the details of this method are still evolving to form a set of software. It is used for aerodynamic & aerothermal quick calculation and analysis of complex geometries with complicated flight status.

References:

[1] Lü Lili, Zhang Weiwei, Ye Zhengyin. Predicting heating distributions for hypersonic reentry bodies [J]. Chinese Journal of Applied Mechanics, 2006,23(2):259-262. (in Chinese)

[2] Han Dong, Fang Lei. Study of aerodynamic heating predictions for hypersonic aircrafts by tracking the surface stream trace[J]. Journal of Aerospace Power, 2009,24(1):65-69. (in Chinese)

[3] Hideyuki T, Kodera M, Tomoynki K, et al. Aeroheating measurements on a reentry capsule model in free-piston shock tunnel HIEST[R]. AIAA 2010-1181, 2010.

[4] Liu Chen, Wang Jiangfeng, Wu Yizhao. Extending Van Leer scheme for flows at all speeds[J]. Chinese Journal of Computational Mechanics, 2001, 28(2): 205-209. (in Chinese)

[5] Pan Sha, Tian Zhengyu, Feng Dinghua, et al. Computation and analysis of aeroheating on hypersonic inlet leading edye[R]. AIAA 2009-7370, 2009.

[6] Eckert R G. Engineering relations for friction and heat transfer to surface in high velocity flow[J]. J Aeronaut Sci, 1956,23(8):585-587.

[7] Zoby E V, Moss J N, Sutton K. Approximate convective heating equations for hypersonic flows[R]. AIAA 79-1078, 1979.

[8] Lees L. Laminar heat transfer over blunt nosed bodies at hypersonic flight speeds[J]. AIAA Journal, 2003,41(7):356-366.

[9] Zhang Zhicheng. Hypersonic aerodynamic heating and thermo-protection[M]. Beijing: National Defence Industry Press, 2006. (in Chinese)

[10] Liou M S. A sequel to AUSM: AUSM^{+}[J]. J Computational Physics, 1996,129:364-382.

[11] Fay J A, Riddle F R. Theory of stagnation point heat transfer in dissociated air[J]. Journal of the Aeronautical Science, 1958,25(2):73-85.

[12] Yang Kai, Gao Xiaowei. Engineering algorithm for aeroheating enviroment of hypersonic aircrafts[J]. Missiles and Space Vehicles, 2010(4): 19-23. (in Chinese)

(Executive editor: Zhang Huangqun)