2−6

Hypersonic Flow Analysys around OREX and Hyperboloid Flare

Takuji Kishimoto
Gifu Technical Institute,
Kawasaki Heavy Industries, LTD.

Shinichi Kaneko
Aerospace Engineering Division,
Kawasaki Heavy Industries, LTD.

ABSTRACT

We have carried out hypersonic flow analysis around OREX(Orbiter Reentry Experiment) and hyperboloid flare by solving axi-symmetric Navier-Stokes Equations. Our flow solver is based on a finite volume implicit TVD upwind scheme. Convective fluxes are evaluated by AUSMDV scheme in order to remove "carbuncle phenomena", that are unphysical solution often appeared on hypersonic flow analysis around a blunt body. We also adopted a curve fitting to estimate thermodynamic and transport properties of an equilibrium air.

Flow condition for OREX is correspond to a flight condition at altitude 56.9km, where real gas effect should not be ignored. Therefore, we assumed this flow not only to be a perfect gas, but to be an equilibrium air.

In the case of hyperboloid flare, flow condition is correspond to experiment by Gottingen Ludwig Tube used cold gas. Numerical result of this problem suggests us that the flow unsteadiness must be taken into account, in order to predict accurately the aerodynamic and aerothermodynamic characteristics of a hypersonic flow including seperations induced by the interaction between a shock wave and a boundary layer.

1. Introduction

In Japan, HOPE(H-II Orbiting Plane) development program has been proceeding by NAL(National Aerospace Laboratory) and NASDA(National Space Development Agency of Japan). In a design of such a winged reentry vehicle, aerodynamic heating in a hypersonic flight regime is one of some critical points over a wide range of flight speed.

In order to investigate the characteristics of aerodynamic heating in a hypersonic flow with "real gas effects", we have carried out some wind tunnel tests using a high enthalpy shock tunnel in recent years[1]. But we cannot depend only on high enthalpy shock tunnel tests to get all aerodynamic heating data that we need, because the technique of high enthalpy shock tunnel testing are not matured yet, especially in the decision of free stream conditions and the reproducibility of test data.

Therefore, CFD is required to supplement the wind tunnel test data. In order that CFD fill this role, it is indispensable that the validation for the wind tunnel test data or the flight data have been carried out sufficiently, because CFD technique also are not matured yet, especially in numerical physical/chemical models.

Simultaneously with the validation of our hypersonic flow analysis system, we have participated in this "High Enthalpy Flow Workshop" and carried out hypersonic flow analysis around OREX(Orbital Reentry Experiment) and a hyperboloid flare with/without real gas effects. We will show the outline of our numerical results as follows .

2. Numerical Approaches [2, 3]

Our flow solver for a perfect gas and an equilibrium air is based on finite volume TVD upwind scheme for thin layer Navier-Stokes equations as basic equation.

MUSCL type TVD scheme is adopted for descretization in space. This scheme limits the variation of characteristic variables by MINMOD function in each cell. Our flow solver has a second order accuracy in space by using this method.

Convective fluxes are evaluated by AUSMDV scheme[4]. This method is one of AUSM type splitting schemes[5] , and has equal simplicity and robustness to flux vector splitting schemes and equal resolution to flux difference splitting schemes. By using AUSMDV scheme, carbuncle phenomena, that are unphysical solution often appeared on a hypersonic flow analysis around a blunt body, can be suppressed to some extent.

Time Integration is implicitly executed by planar Gauss-Seidel relaxatiom method.

In the case of an equilibrium air, thermodynamic and transport properties are calculated by a curve fitting method proposed by Dr.Srinivasan et al[6, 7].

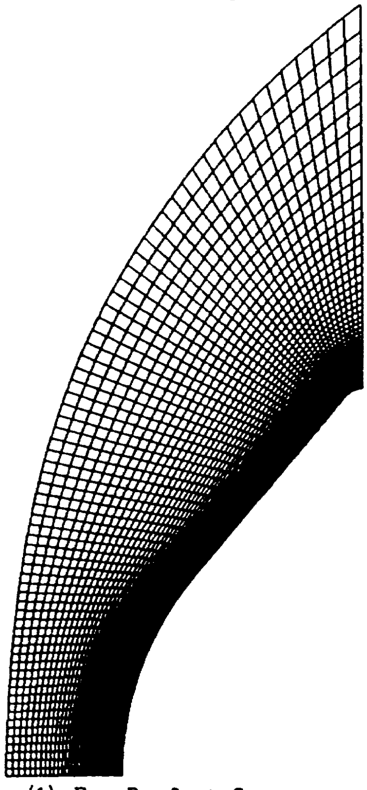
3. Numerical Results

Our numerical results as follows assume a laminar flow for either OREX or a hyperboloid flare.

(1) OREX

Our numerical results for OREX are solutions to Problem Ⅱ.4(chemical equilibrium) and Ⅱ.5(perfect gas) of this workshop, respectively.

Sizes of computational grids as we used are 71×81 for a perfect gas case, and 66×41 for an equilibrium air case(Fig.1).


(1) For Perfect Gas

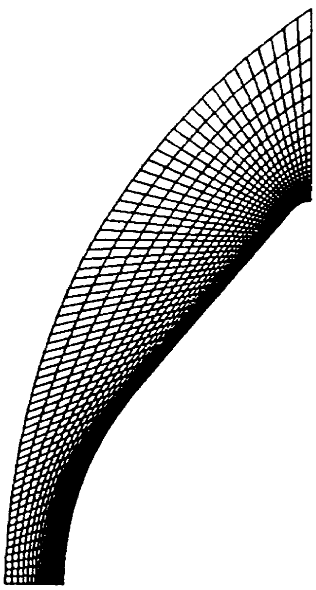

(2) For Equilibrium Air

Fig.1 Computational Grids around OREX

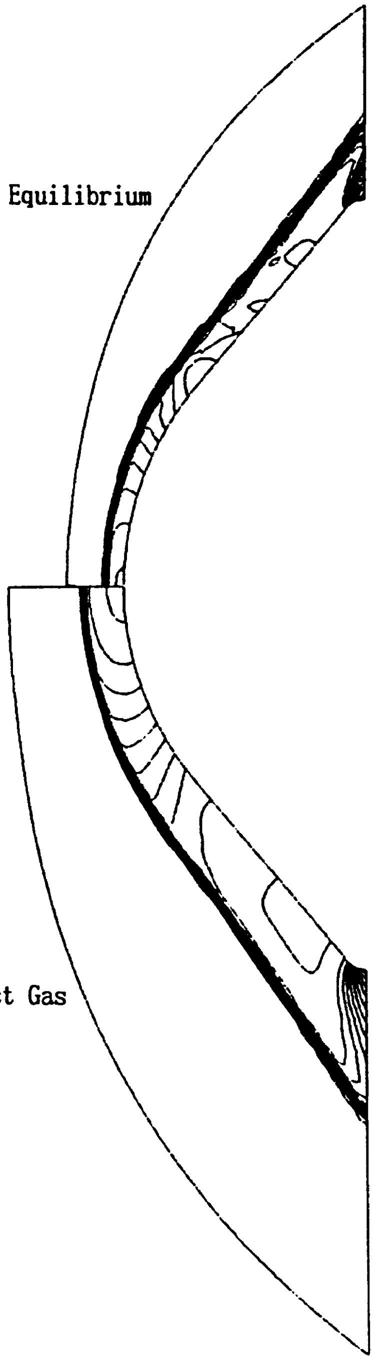


Fig. 2 Pressure Contours around OREX

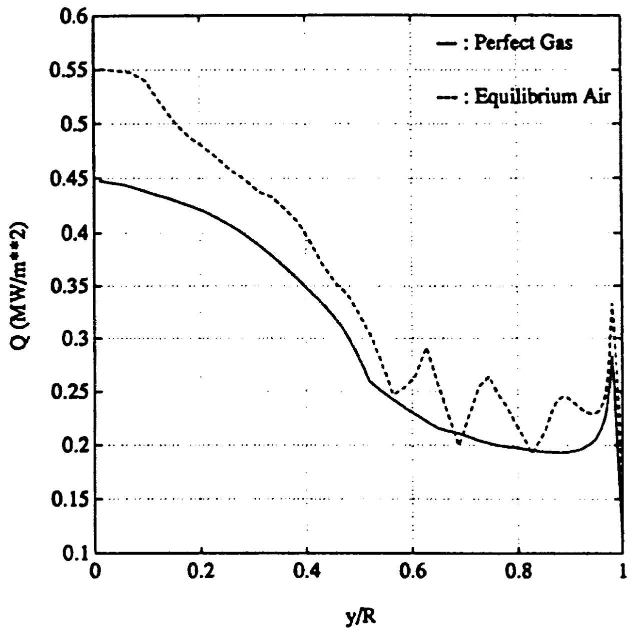


Fig. 3 Heat Flux Distribution on OREX

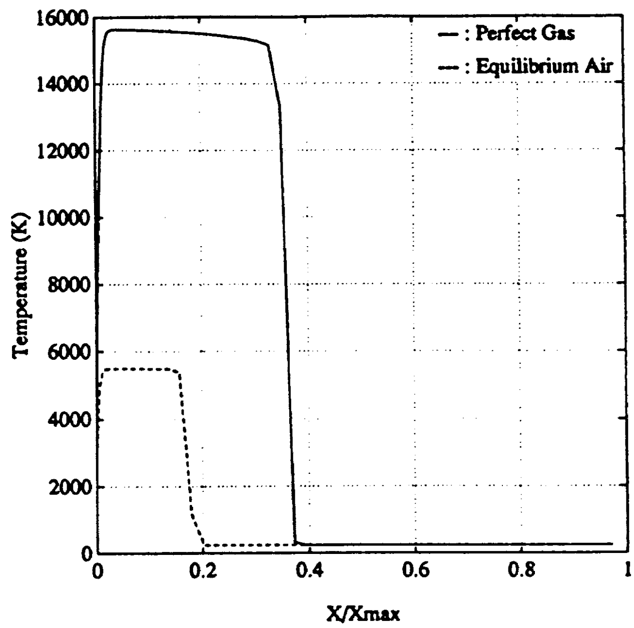


Fig. 4 Temperature Distribution on Stagnation Stream Line of OREX

Flow condition, which is correspond to flight condition at altitude 56.9km in a Japanese first successful reentry experiment from orbit, are as follows.

Velocity	:	5562 m/s
Temperature	:	248.1 K
Pressure	:	23.60 Pa
Wall Temperature	:	1519 K

In these results, the characteristic difference between a perfect gas and an equilibrium air is that surface heat flux of an equilibrium air is higher than of a perfect gas (Fig. 3), though a temperature behind a bow shock of an equilibrium air is about 10,000K lower than of a perfect gas (Fig. 4). This difference of heat flux is led by recombination, in which reaction heat is released, near solid surface in an equilibrium air, which is regarded as a fully catalytic wall in our numerical simulation.

With respect to the bow shock location, the result of an equilibrium air case is about a half distance from a body of a perfect gas case(Fig.4). In the view of this aspect, these numerical results show that our hypersonic flow analysis system can definitely capture the real gas effects in chemical equilibrium.

But heat flux distribution on surface shows unexpected oscillation in an equilibrium air case(Fig.3). Therefore, our next goal of hypersonic flow analysis is that such unphysical oscillation in an equilibrium air flow should be removed.

(2) Hyperboloid Flare

Our numerical result for a hyperboloid flare is solution to Problem III-1 of this workshop, that is only a perfect gas case. This problem is correspond to an experiment by Gottingen Ludwig Tube used cold gas.

Computational grid as we used is generated by Dr.Y.Yamamoto(NAL), and a size of this grid is 521×101(Fig.5).

Flow conditions are as follows.

Mach Number	:	6.83
Temperature	:	67.765 K
Reynolds Number	:	7.0×10^{6} /m
Wall Temperature	:	300 K

In a heat flux distribution on a surface (Fig.7), negative values appear from x/L=0.8 to 1.0, where is just before and behind a peak value at a reattached region. This may be because we do not take account of the unsteadiness owing to a seperation induced by the interaction between an oblique shock wave and a boundary layer in this calculation which is used local time stepping to get a steady solution.

Such influences of the lack of the flow unsteadiness in numerical simulation also appears as a small oscillation in temperature contours(Fig.6) and a pressure distribution on a surface (Fig.8).

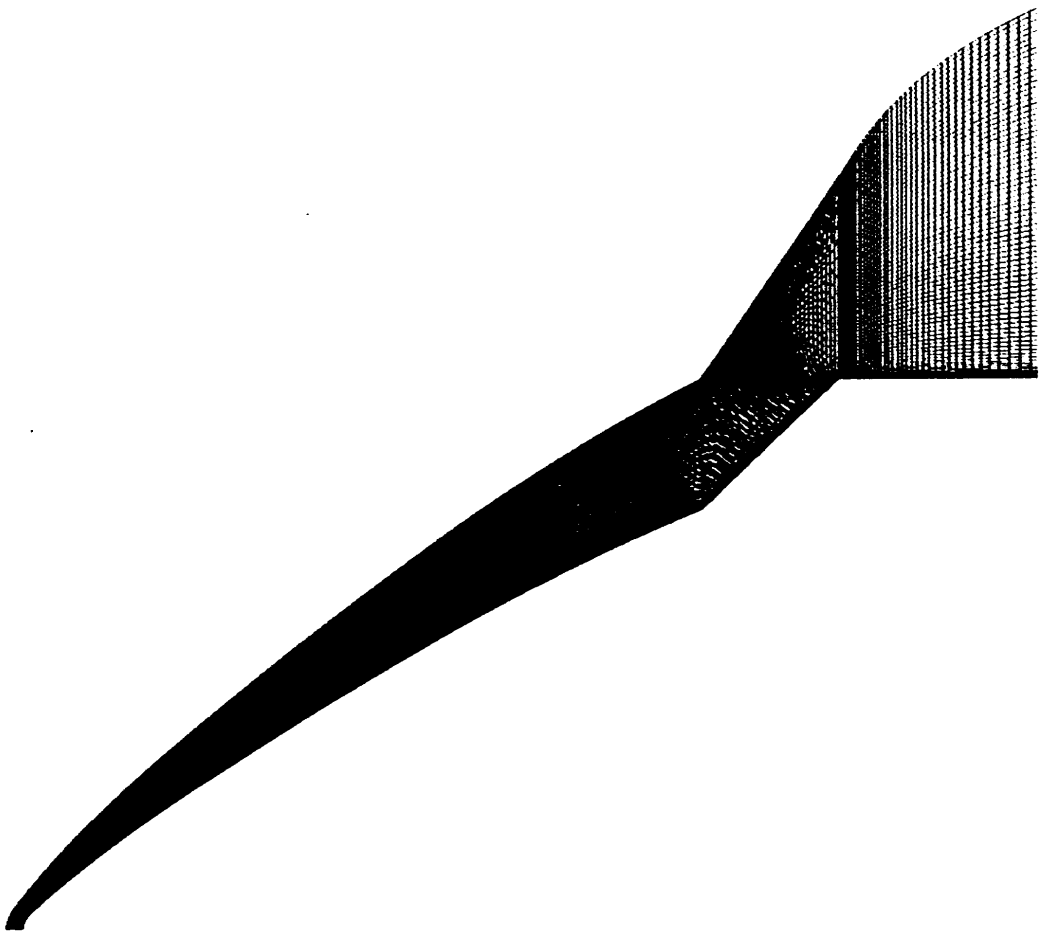


Fig.5 Computational Grid around Hyperboloid Flare

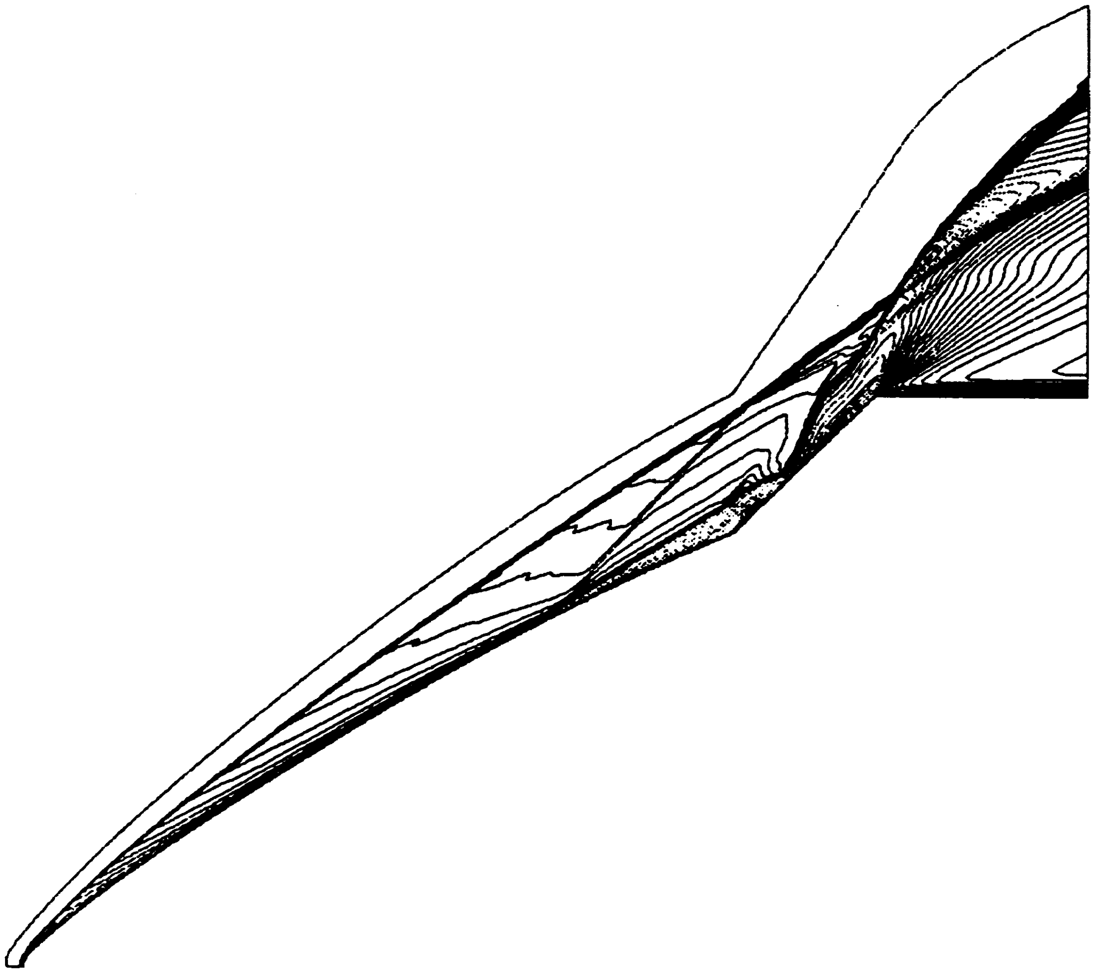


Fig. 6 Temperature Contures around Hyperboloid Flare

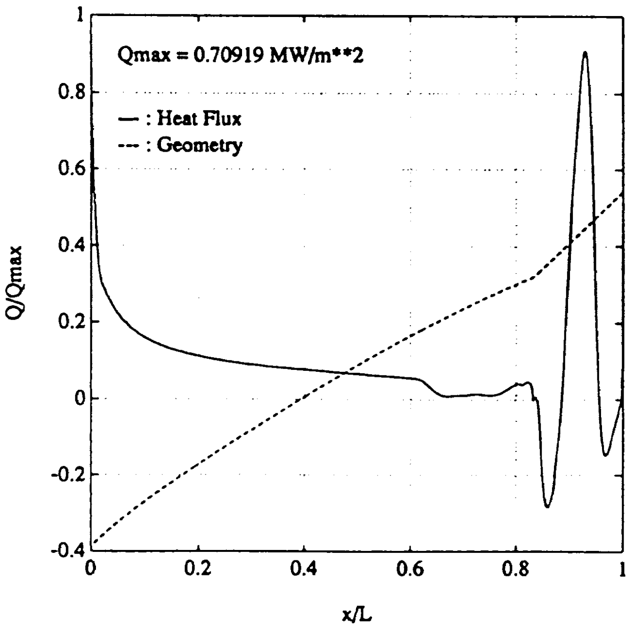


Fig. 7 Heat Flux Distribution on Hyperboloid Flare

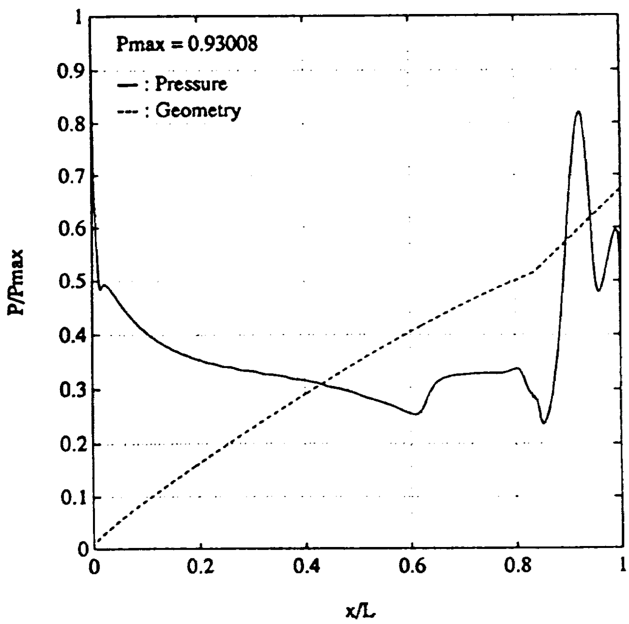


Fig. 8 Pressure Distribution on Hyperboloid Flare

4. Conclusions

We introduced our numerical results of hypersonic flow analysis around OREX and hyperboloid flare with/without real gas effects.

Numerical results of OREX suggest us that our flow analysis system for a hypersonic flow is not completed yet, especially for an equilbrium air flow. The result of a hyperboloid flare also shows that the flow unstediness must be taken into account, in order to predict more accurately the aerodynamic and aerothermodynamic characteristics of a hypersonic flow including seperations induced by the interaction between a shock wave and a boundary layer.

Therefore, we intend to make our system more accurate and usable, and our final goal is that our flow analysis system can be used as one of aerodynamic and aerothermodynamic design tools for the development of reentry vehicles.

References

(1) Hanamitsu, A., Kishimoto, T., and Bito, H., "High Enthalpy Flow Computation and Experiment around the Simple Bodies", Proceedings of the 13th NAL Symposium on Aircraft Computational Aerodynamics, to be appeared.

(2) Sawada, K., and Takanashi, S., "A Numerical Investigation on Wing/Nacelle Interferences of USB Configuration", AIAA Paper 87-0455, 1987.

(3) Kishimoto, T., and Kaneko, S., "Hypersonic Flow Analysis around OREX", Proceedings of the 12th NAL Symposium on Aircraft Computational Aerodynamics, CFD Workshop on Hypersonic Flow, NAL SP-26, pp.56-60, 1994. In Japanese.

(4) Wada, Y., and Liou, M.-S., "A Flux Splitting Scheme with High-Resolution and Robustness for Discontinuities", AIAA Paper 94-0083, 1994.

(5) Shima, E., and Jounouchi, T., "Role of Computational Fluid Dynamics in Aeronautical Engineering (No.12) -Formulation and Verification of Uni-Particle Upwind Schemes for the Euler Eqations-", Proceedings of the 12th NAL Symposium on Aircraft Computational Aerodynamics, NAL SP-27, pp.255-260, 1994. In Japanese.

(6) Srinivasan, S., Tannehill, J.C., and Weilmuenstar, K.J., "Simplified Curve Fits for the Thermodynamic Properties of Equilibrium Air", ISU-ERI-Ames-86401, ERI Project 1626, CFD 15, 1986.

(7) Srinivasan, S., and Tannehill, J.C., "Simplified Curve Fits for the Transport Properties of Equilibrium Air", NASA CR-178411, 1987.