Numerical and Experimental Analysis of Transpiration Cooling in Carbon Dioxide Atmosphere at Hypersonic Mach Numbers

V. N. Kulkarni, P. S. Kulkarni and K. P. J. Reddy
Department of Aerospace Engineering, Indian Institute of Science, Bangalore-560 012, India
Email: psk@aero.iisc.ernet.in

ABSTRACT

The numerical solutions are obtained for skin friction, heat transfer to the wall and growth of boundary layer along the flat plate by employing two dimensional Navier–Stokes equations governing the hypersonic flow coupled with species continuity equations. Flow fields have been computed along the flat plate in CO_2 atmosphere in the presence of transpiration cooling using air and carbon dioxide.

1. INTRODUCTION:

The main focus of hypersonics research in recent times is on the re-entry aerodynamics in order to develop protection system for the vehicles from severe thermal loads encountered during this phase. The aerodynamic heating is a major design concern for vehicles flying at hypersonic Mach numbers. The thermal load problem is handled by using an effective cooling system. The transpiration cooling in which a cold fluid is injected into the boundary layer through a porous media such that coolant issues forth as a continuous mass is an important cooling technique. Many experimental[1] as well as analytical and numerical studies of transpiration cooling have been reported in the literature[2,3]. Complete Navier-Stokes solutions in two-dimensional form for transpiration cooling in earth's atmosphere using air, helium and carbon dioxide as coolants have been reported recently using CFD techniques[3-5]. The focus in recent times has been on the planet Mars and many probes have been sent to explore the life on Mars. This paper presents an analysis of transpiration cooling in carbon dioxide atmosphere by considering a simple flat plate geometry flying at hypersonic Mach number and the transpiration cooling is achieved by injecting air and carbon dioxide gases. The N-S equations governing the hypersonic flow coupled with species continuity equations are solved numerically using finite volume technique. The numerical results generated using this CFD code have been verified and also validated by conducting experiments in hypersonic shock tunnel of IISc.

2. GOVERNING EQUATIONS AND NUMERICAL ALGORITHM:

Assuming CO_2 as the test gas flowing at hypersonic Mach number while the transpiration cooling is achieved by injecting a coolant gas the basic Navier-Stokes (N-S) equations are modified with appropriate species continuity equations. The modified non-dimensional N-S equations in the conservative integral form are written as:

$$\frac{\partial \overline{U}}{\partial t} + \frac{1}{A}\oint_S \vec{F}\,\vec{n}\,dS = 0 \qquad (1)$$

Where, $\overline{U} = \frac{1}{A}\int\int U dA$ is the average value of the column vector of conserved variables $U = \left[\rho, \rho u, \rho v, \rho E, \rho m_1\right]^T$ in the control volume and $\vec{F}$ is the flux vector defined in Ref. [4]. Here, ρ is the density, u and v are the velocity components in the x and y directions, E is the total energy of the mixture, A and S are the area and boundary of the control volume, m_1 is the mass fraction of the species 1and $\vec{n}$ is the unit outward normal. The viscosity coefficient μ for the gas mixture is expressed as a function of temperature based on either Sutherland's law or Wilkes mixture rule[3].

Report Documentation Page

Form Approved
OMB No. 0704-0188

1. REPORT DATE **14 APR 2005**	2. REPORT TYPE **N/A**	3. DATES COVERED **-**

4. TITLE AND SUBTITLE **Numerical and Experimental Analysis of Transpiration Cooling in Carbon Dioxide Atmosphere at Hypersonic Mach Numbers**	5a. CONTRACT NUMBER 5b. GRANT NUMBER 5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S)	5d. PROJECT NUMBER 5e. TASK NUMBER 5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **epartment of Aerospace Engineering, Indian Institute of Science, Bangalore-560 012, India**	8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR'S ACRONYM(S) 11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release, distribution unlimited

13. SUPPLEMENTARY NOTES
See also ADM001800, Asian Computational Fluid Dynamics Conference (5th) Held in Busan, Korea on October 27-30, 2003., The original document contains color images.

14. ABSTRACT

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT **unclassified**	b. ABSTRACT **unclassified**	c. THIS PAGE **unclassified**	**UU**	**3**	

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std Z39-18

The CFD code developed is based on the finite volume method. Discretising and applying to a cell (i, j) the above equations lead to:

$$\frac{\partial U_{ij}}{\partial t}+\frac{1}{A_{ij}}\left\{\sum_{l=1}^{4}\vec{F}.\vec{n}\,ds\right\}_{ij}=0 \qquad (2)$$

In the present work, Roe's scheme is used to determine the inviscid fluxes. The quantities such as $u,\ v,\ T, \partial u/\partial x$, etc., are required on an interface in order to evaluate the viscous fluxes. In the evaluation of the derivatives like $\partial u/\partial x$ and $\partial u/\partial y$ at an interface the auxiliary cell approach is used. The resulting set of equations is treated with the Lind-Jacobi relaxation algorithm.

3. RESULTS AND DISCUSSIONS:

The code is used to compute flow fields at M_∞=6 &7 and Re = 10^6 and 5X 10^6. The variation of normalized skin friction coefficient along the flat plate for M_∞=6 and Re = 10^6 for both the injecting gases for different blowing parameter is shown in Fig 1. The CO_2 is found to be more effective in reducing the skin friction. The variation of temperature in the normal direction and the variation in wall temperature for the same Mach number and Reynolds number are shown in Figs 2 &3. The results clearly show that the thermal boundary layer gets affected significantly due to blowing. It is also clear that the injecting gas reduces the wall temperature significantly. The experimental investigations have also been carried out to validate the CFD results and will be presented in the full length paper.

REFERENCES

[1] Libby PA, Cresci RJ. Experimental investigation of the downstream influence of stagnation point mass transfer. Journal of Aeronautical Sciences, 1961, 28: 51-64

[2] Hartnett JP, Eckert ERG. Mass transfer cooling in a laminar boundary layer with constant fluid properties. Trans. Amer. Society for Mechanical Engineers, 1957, 79: 247

[3] Sreekanth, Reddy NM. Transpiration cooling analysis at hypersonic Mach numbers using Navier-Stokes equations. AIAA Paper 94-2075 (1994)

[4] Ravi BR. Transpiration cooling analysis including binary diffusion using 2-D Navier-Stokes equations at hypersonic Mach numbers. MS dissertation, Dept. of Aero Engg., IISc, Bangalore, India (1997).

[5] PS Kulkarni, Ravi BR and KPJ Reddy "Two dimensional Navier-Stokes Solutions for transpiration cooling at hypersonic Mach numbers" 9th ACFM, IRAN

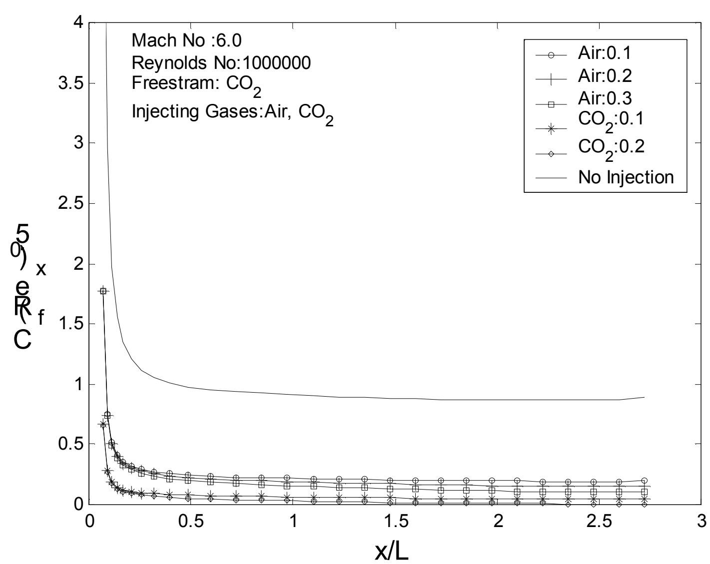

Fig 1. variation of skin friction for injection of different gases for
M = 6.0 and R = 10 6

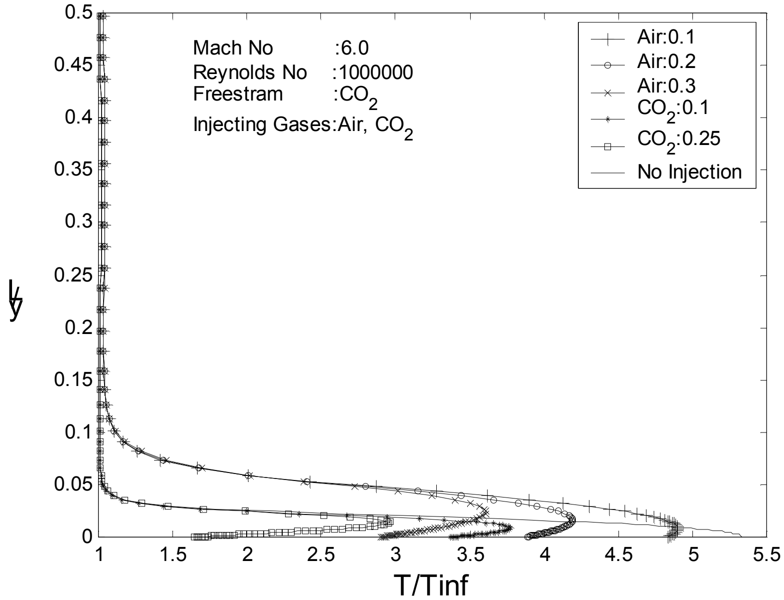

Fig2. variation of temperature profile in normal direction for
M = 6.0 and R = 10 6

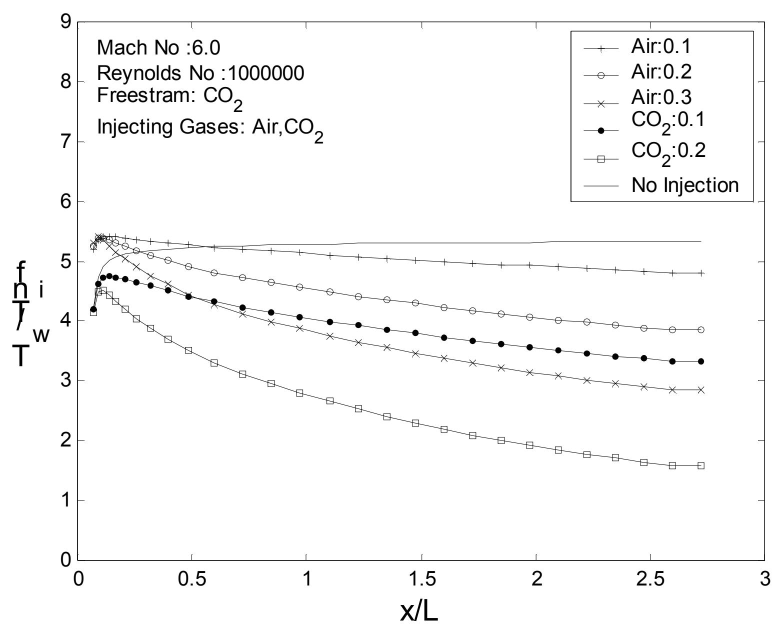

Fig3. Variation in wall temperature for different injecting gases for
M = 6.0 and R = 10 6