© 2018 IJEDR | Volume 6, Issue 1 | ISSN: 2321-9939

Numerical Flow Simulation over a Sphere Cone Reentry for Different Atmospheric Conditions

[1] Manigandan P, [2] Vadivelu P, [3] Naveen R
[1,2,3] Assistant Professor
[1,2,3] Department of Aeronautical Engineering,
[1,2,3] Bannari Amman Institute of Technology, Sathyamangalam, India

***Abstract*— Flow simulations are carried on two different atmospheric conditions like air and carbon-di-oxide, provided for the same pressure, temperature and Mach number using commercial flow solvers. The re-entry atmosphere for Earth is air and for other exoplanets like Mars and Venus, the atmosphere mainly consists of carbon-di-oxide. The main objective of the paper is to present the effect of pressure, temperature and Mach number upon the aerodynamic characteristics of Sphere cone shaped re-entry capsule for the given different atmospheric conditions. The computational study in this paper demonstrates the importance of shock waves and explains the changes in the flow field for similar boundary conditions and variations in all aspects for these different atmospheric environments.**

***Index Terms*— Re-Entry, Sphere cone, Bow shock wave, Air, Carbon di oxide, Mach number.**

I. INTRODUCTION

Atmospheric entry is the movement of an object through space in vacuum medium, which gradually enters into the atmospheres of Earth and other exoplanets in the space. Re-entry vehicle is a part of the spacecraft, which is designed to re-enter the atmosphere of planets to carry out and return, after successful space missions. In this paper, the re-entry vehicle is a type of Sphere cone re-entry capsule that has a large flat surface frontal area in the fore-body with a short rear body length to reduce ballistic coefficient. In future, we may have to bring back the rovers to Earth and in-order to do so we have to take into consideration the materials that can operate and withstand both adverse conditions [1].

Hypersonic flow over the nose section creates a detached bow shock which results high surface pressure and high aerodynamic drag. It has mixed subsonic and supersonic region between the bow shock and fore body. This investigation considers Mach 3.0 and Mach 5.0 for analyzing the Temperature, Pressure and Mach number variations [2]–[4]. The analyzed variations in Pressure, Temperature and Mach number are then compared with each other at the given Mach numbers and operating conditions.

II. METHODOLOGY

Geometrical Model and Computational model

Ballistic probes have an advantage that they do not require any guidance but can have control surfaces like flaps to induce drag in some special cases. A re-entry vehicle usually has a flat fore-body with large surface area to inscribe high drag coefficient.

A sphere-cone model is used to carry out analysis in different simulating atmospheric conditions. The sphere-cone is a spherical section with a frustum or blunt cone attached. The sphere-cone's dynamic stability is typically better than that of a spherical section. The angle between cone's axis and the surface is termed as half cone angle. It plays an important role in aerodynamic forces acting on a re-entry body [4], [5].

Computational Fluid Dynamics is a software tool used to analyses fluid flow, heat transfer and chemical reactions. CFD analysis is generally divided into three categories such as pre-processing, solver and post-processing. A solver used to perform analytical methods undergoes the following steps such as approximation of flow variables and defining the models, materials and cell zone conditions where discretization of governing and energy equations are taken into consideration. At last, algebraic equations are used to solve the given input conditions. The type of solver taken into consideration is Pressure-based solver and time as steady. Velocity formulation is taken as absolute in 2D space planar [6]–[8].

Energy equation is taken into consideration. The viscous medium is taken with Realizable K-ε model is used, Non-Equilibrium wall function is opted for near wall treatment. The material selected for density is having ideal gas properties and the selected viscosity is Sutherland's. Implicit scheme is chosen and coupled is used for flux type with discretization done at least square cell based with second order upwind for flow energy equation. The courant number is set to 100 for Air atmosphere and 75 for CO_2 atmosphere and hybrid initialization is done [6], [9], [10]. Post-processing is the stage where the domain geometry and grid display with flows inside and around the body of the object is displayed, 2D plots, vector plotting and line and shaded contour plots are displayed. Particle tracking is done to visualize the flow of a particle and to check whether it swirls or changes direction around the object.

Boundary Conditions

Fluent is used to analyze the flow simulations over the re-entry body before going for experimental testing. The boundary conditions given for the re-entry body considering both the atmospheres are as follows.

Table 1 Boundary conditions

Parts	**Boundary Conditions**
Inlet	Pressure far-field
Far-field	Pressure far-field
Sphere cone	Wall
Symmetry	Symmetry
Outlet	Pressure outlet

The re-entry sphere cone capsule was examined in the free stream condition at an altitude of 76 km and 85 km flight condition in Air atmosphere and the same inputs of pressure and temperature is given for CO_2 atmosphere to draw out a comparison with operating condition of pressure as zero Pascal.

Table 2 Free stream boundary conditions

M_∞	**P_∞, Pa**	**T_∞, K**
3.0	2073	224
5.0	1238	232

III. Results and Comparison

Figure 1 Static pressure contour at Mach 3 in Air and CO_2 respectively

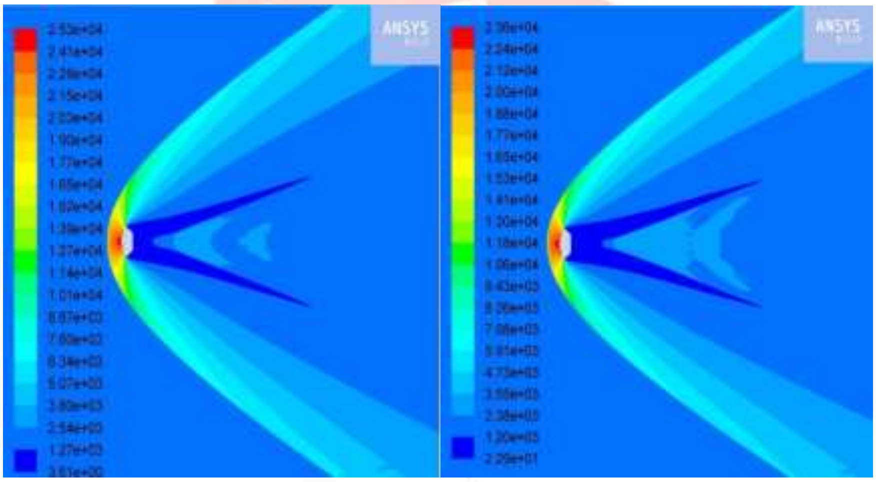


Figure 2 Static temperature contour at Mach 3 in Air and CO_2 respectively

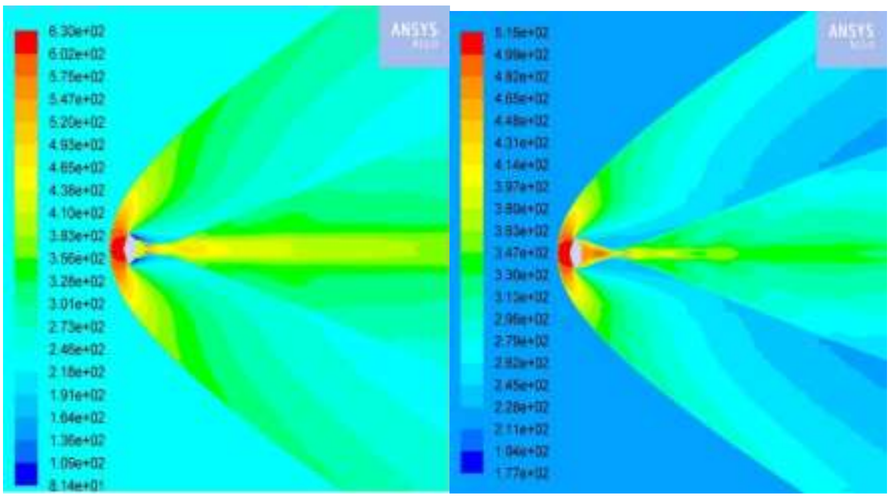
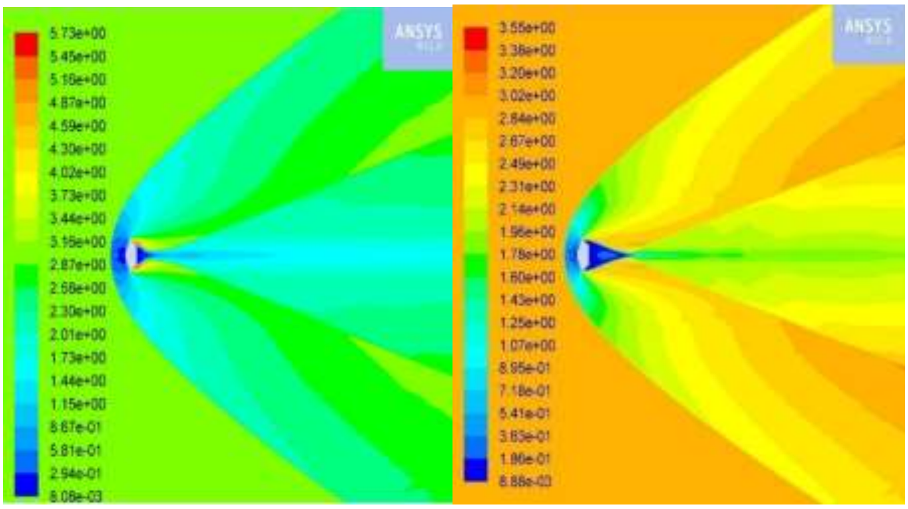


Figure 3 Mach number contour at Mach 3 in Air and CO_2 respectively



From the above inferences, it is made clear that the Mach number, velocity, density, turbulence and temperature are higher in air atmosphere when it is compared to carbon-dioxide atmosphere. Shock wave studies have revealed that the fore-body of the re-entry capsule

Figure 4 Static pressure contour at Mach 5 in Air and CO_2 respectively

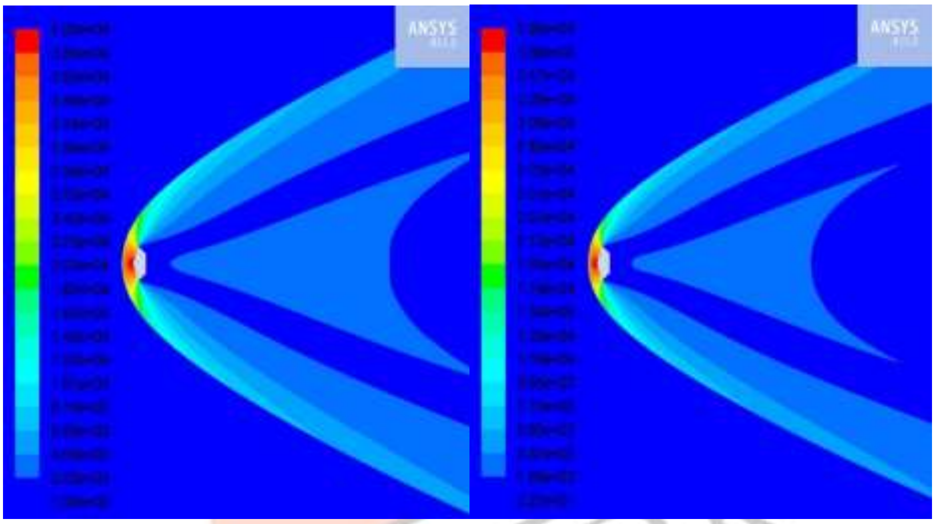


Figure 5 Static temperature contour at Mach 5 in Air and CO_2 respectively

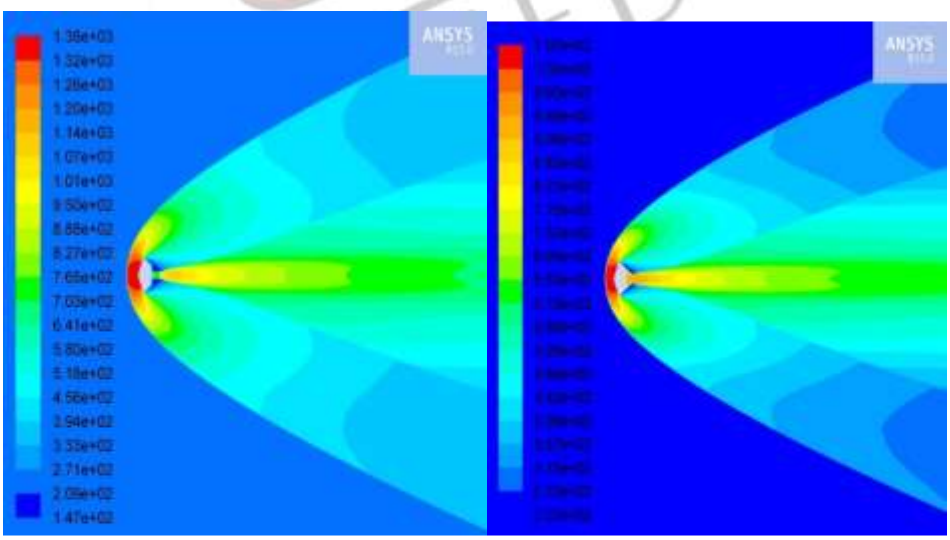
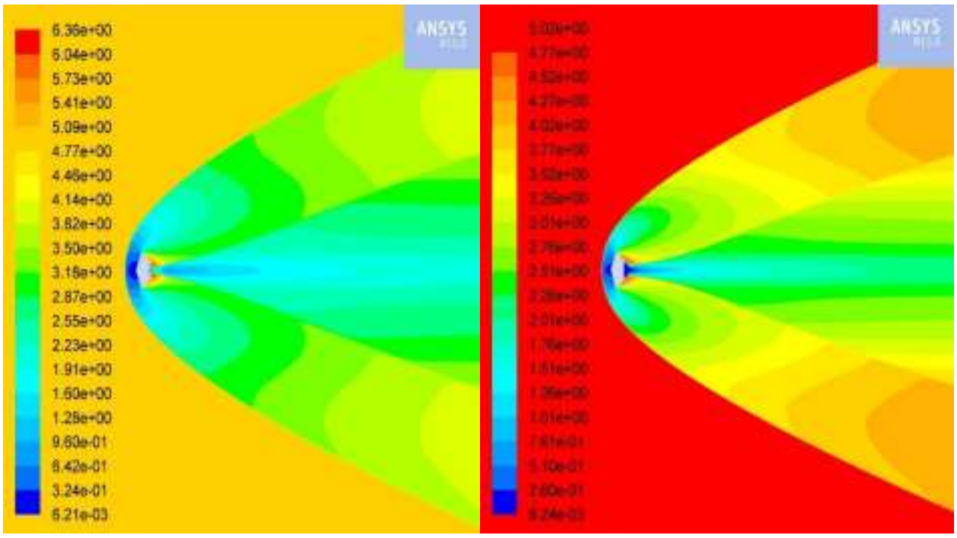


Figure 6 Mach number contour at Mach 5 in Air and CO_2 respectively



From the above inferences, we can see that the distance between the shock wave and object decreases as the Mach number increases.

Comparison of Graphs

Figure 7 Mach number variation along X direction at Mach 3

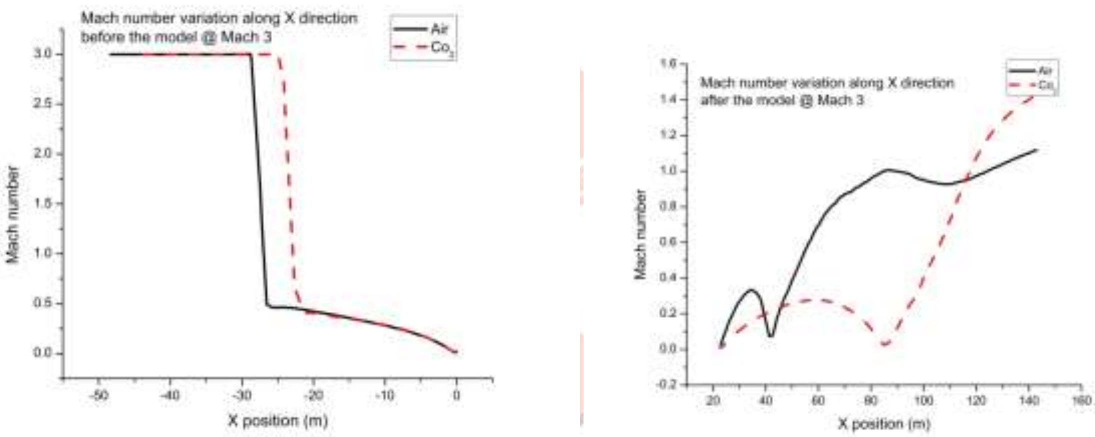


Figure 8 Mach number variation along X direction at Mach 5

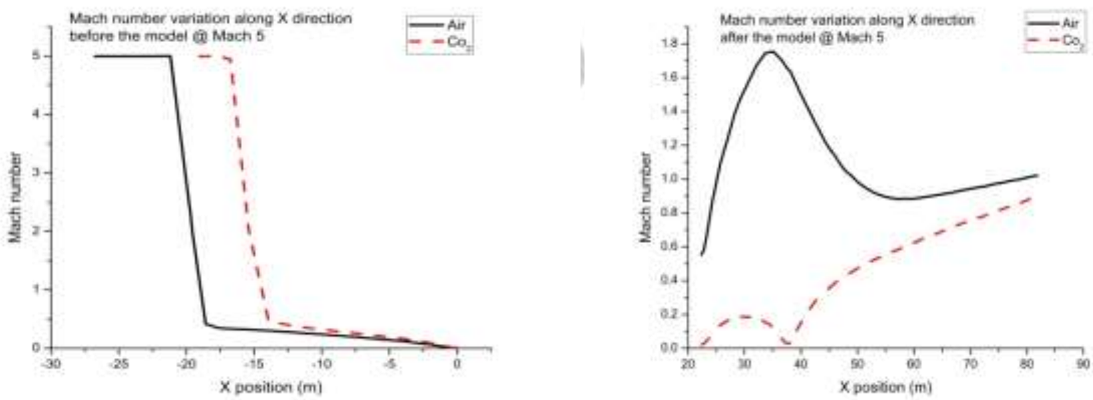


Figure 9 Static temperature variation along X direction at Mach 3

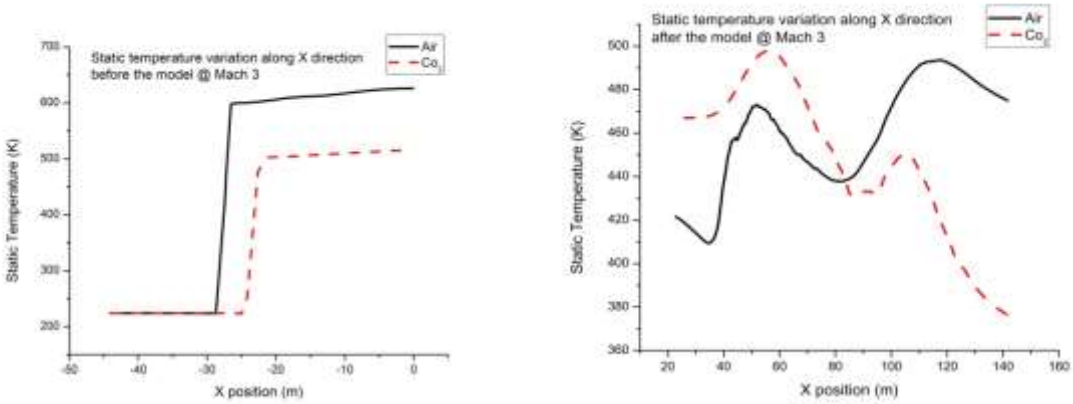


Figure 10 Static temperature variation along X direction at Mach 5

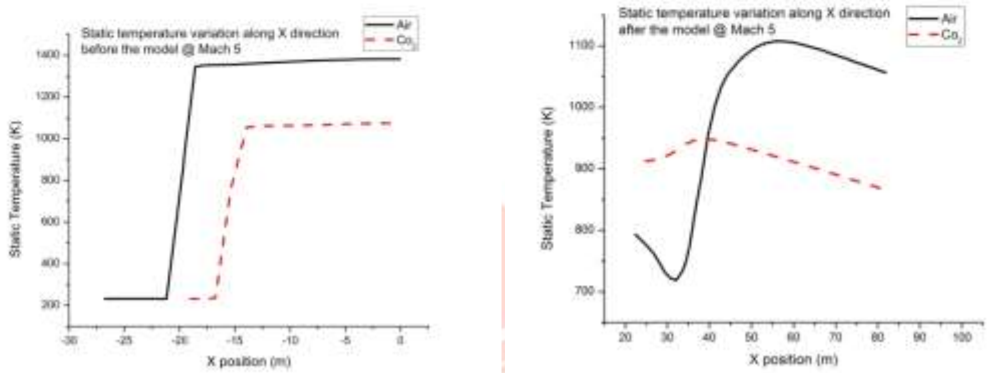


Figure 11 Static pressure variation along X direction at Mach 3

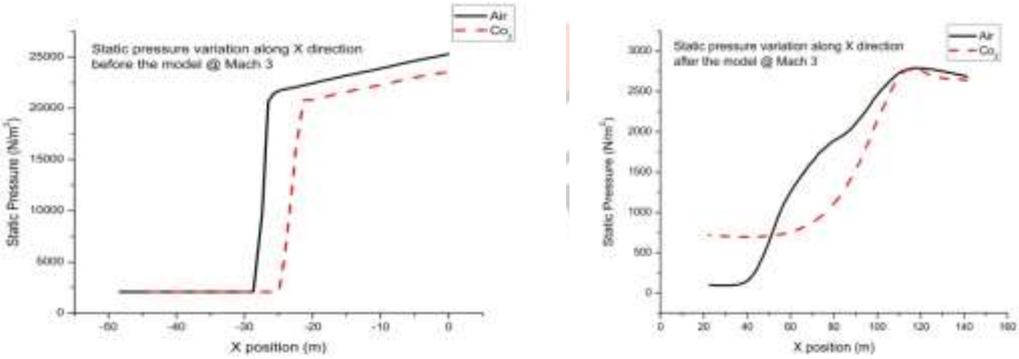


Figure 12 Static pressure variation along X direction at Mach 5

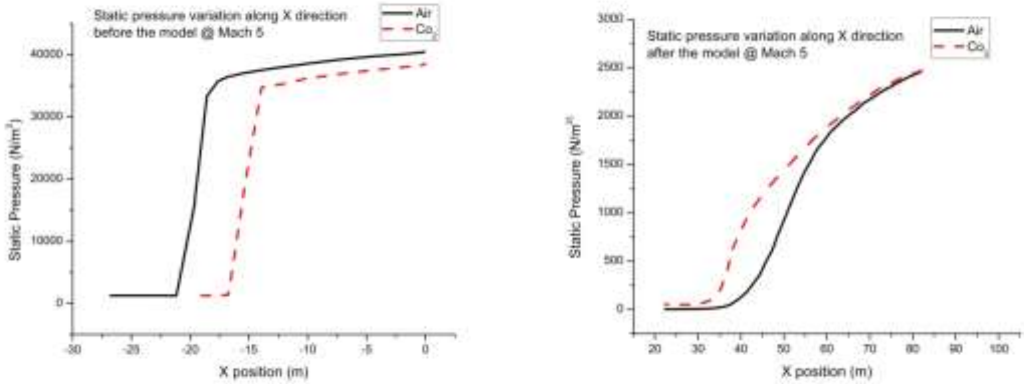


Table 3 Drag Values

Mach Number	Atmospheric Condition	Pressure Drag (N)	Viscous Drag (N)	Total Drag (N)
Mach 3	Air	567104.45	114.31871	567218.76
	Co_2	517998.74	82.829918	518081.57
Mach 5	Air	920169.07	204.05893	920373.13
	Co_2	868458.56	172.74672	868631.31

IV. CONCLUSION

Analysis have been carried out for two simulating atmospheric conditions such as air and carbon-di-oxide and the results have been found out to be contrasting. Characteristic flow features around the blunt body at supersonic and hypersonic speeds are observed. The variations in velocity, pressure and temperature for different atmospheric conditions at different Mach numbers have been plotted in graphs. It has been observed that Mach number behind the shockwave have been decreased gradually and have even become zero at stagnation point. Results adequately explains that in a situation where air is used there is a high surface pressure on the fore-body which results in the development of high aerodynamic drag when compared to carbon-di-oxide. Similarly, the temperature is also observed to be higher in air atmosphere than the carbon-di-oxide atmosphere. The velocity in terms of Mach number behind the model is also observed to be higher in air atmosphere than the carbon-di-oxide atmosphere.

REFERENCES

[1] R. D. Braun and R. M. Manning, “Mars Exploration Entry, Descent, and Landing Challenges,” J. Spacecr. Rockets, vol. 44, no. 2, pp. 310–323, 2007.

[2] D. L. Shultz, T. V. V. T. Jones, D. L. Schultz, and T. V. V. T. Jones, “Heat Transfer measurement in short duration hypersonic facilities,” 1973.

[3] B. R. Hollis, “User’s Manual for the One-Dimensional Hypersonic Experimental Data Reduction Code,” no. Contractor report 4691, 1995.

[4] B. R. Hollis, J. N. Perkins, and N. Orleans, “AIAA 96-2240 Hypervelocity Heat-Transfer Measurements in an Expansion Tube Raleigh , NC 19th AIAA Advanced Measurement and Ground Testing Technology Conference,” 1996.

[5] D. N. Kendall, W. P. Dixon, and E. H. Schulte, “Semiconductor Surface Thermocouples for Determining Heat-Transfer Rates,” IEEE Trans. Aerosp. Electron. Syst., vol. AES-3, no. 4, pp. 596–603, 1967.

[6] M. S. Ivanov and S. F. Gimelshein, “Computational Hypersonic Rarefied Flows,” Annu. Rev. Fluid Mech., vol. 30, no. 1, pp. 469–505, 1998.

[7] R. C. Mehta, “Computations of flow field over Apollo and OREX reentry modules at high speed,” Indian J. Eng. Mater. Sci., vol. 15, no. 6, pp. 459–466, 2008.

[8] R. C. Mehta, “Numerical simulation of supersonic flow past reentry capsules,” Shock Waves, vol. 15, no. 1, pp. 31–41, 2006.

[9] C. J. Roy and F. G. Blottner, “Assessment of One- and Two-Equation Turbulence Models for Hypersonic Transitional Flows,” J. Spacecr. Rockets, vol. 38, no. 5, pp. 699–710, 2001.

[10] P. A. Gnoffo, K. J. Weilmuenster, H. H. Hamilton, D. R. Olynick, and E. Venkatapathy, “Computational Aerothermodynamic Design Issues for Hypersonic Vehicles,” J. Spacecr. Rockets, vol. 36, no. 1, pp. 21–43, 1999.