International Journal of Engineering Research and Development
eISSN : 2278-067X, pISSN : 2278-800X, www.ijerd.com
Volume 2, Issue 4 (July 2012), PP. 29-34

Flow Simulation over Re-Entry Bodies at Supersonic & Hypersonic Speeds

Shiva Prasad U.[1], Srinivas G.[2]
[1,2]Manipal Institute of Technology, Manipal University, Manipal, Karnataka, INDIA.

Abstract— In the present paper, flow simulations are carried on two design configurations of re-entry vehicles, FIRE II and OREX using commercial flow solvers. The purpose of the paper is to present the effect of flight attitude upon the aerodynamic characteristics of an Apollo shaped re-entry capsule. This paper exemplifies the importance of the aerodynamic forces effect on the motion of re-entry vehicle, especially at such high speeds, that even a slight change in angle of attack can severely alter the activity of the re-entry capsule including the shock wave which plays such a big factor in the fate of the craft. The study demonstrates the importance of understanding the effects of shock waves and illustrates how small change in flight attitude can alter the resulting aerodynamic forces on the capsule. Finally, any minor alteration to the shape of the craft will be discussed which have great implications on the dynamics of craft.

Keywords— Capsules, CFD, Mach number, Re-Entry, Shock waves.

I. INTRODUCTION

In this paper, the type of re-entry vehicle considered is re-entry capsule, which re-enters earth's atmosphere from an orbit. The primary design consideration of re-entry capsules requires large spherical nose radius of their fore body that gives high aerodynamic drag and a short body length for reducing the total structural weight and the ballistic coefficient [1]. In order to know more about the planets and the natural satellites present in our Solar system, there is a need for us to enter an orbit around that planet or satellite and observe them. For better understanding, we might even have to send rovers onto the surface of the planet or satellite to conduct experiments, and if possible, bring them back to Earth. For this to happen, the space probe must pass through their atmosphere, reach the surface intact, conduct experiments there, travel back to Earth and perform another re-entry phase in order to reach Earth's surface with the data collected. For this to happen, the re-entry vehicle must be designed accordingly and analysed using experimental and numerical methods which provide valuable knowledge for future spacecrafts such as crew exploration vehicle, especially with the recent call to move back to the old Apollo shaped re-entry vehicles rather than the usage of space shuttles. Hence, the paper explores the analysis on re-entry capsules.

II. OBJECTIVE

The re-entry vehicle considered is a low Ballistic Coefficient (BC) re-entry capsule. In the present work, CFD analysis is carried on two design configurations of re-entry vehicles, FIRE II and OREX using fluent software. A study is taken up on the flow field features around the re-entry vehicle at supersonic and hypersonic speeds.

III. RE-ENTRY CAPSULES

Aeroshells are designed to deliver payloads safely through a planetary atmosphere, protecting the payload from the high aerodynamic heating and loads encountered during EDL (Entry Descent, Landing). An aeroshell generally consists of a forebody which faces the flow and a backshell which completes the encapsulation of the payload. The specific shape of a particular aeroshell is driven by EDL performance requirements and thermal/structural limitations. Here in fig. 1 two different aeroshell shapes are presented.

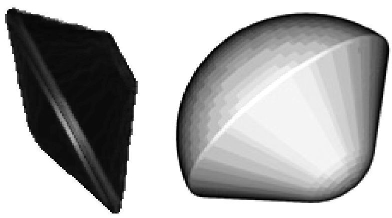

Fig.1 Different Aeroshell Shapes [8,9].

A. Design Configurations

Ballistic probes have the advantage that they do not require guidance and control precautions [4]. Therefore these concepts are less costly than lifting ones but they need low ballistic factors for direct entry. Low ballistic factor means large reference area, high drag coefficient and low mass. Normally, for all known missions, an appropriate mass reduction is critical.

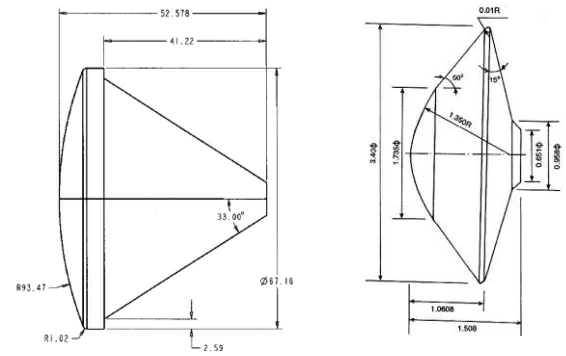

Fig.2 Geometrical details of FIRE II and OREX Re-Entry bodies [2, 3].

Project FIRE was an Apollo era experiment to measure the radiative and convective heating during atmospheric entry at lunar return speeds. The Fire II reentry vehicle consisted of a multi-layer configuration made up of three phenolic-asbestos heat shields sandwiched between beryllium calorimeters. Fig. 2 is a schematic of the vehicle showing the configurations. The 66° included angle conical afterbody section was constructed of a fiberglass shell supporting a layer of phenolic-asbestos heat protection material. A thin surface coating of silicon elastomer was added for pre-launch moisture protection. The conical frustum portion of the afterbody was instrumented with a symmetrical array of 12 gold calorimeters, distributed at three circumferential locations and four stations on the frustum. The calorimeters and their associated heat shields were designed to be ejected after the onset of melting, yielding three separate data-gathering periods.

The OREX geometry is depicted in Fig. 2 with the detailed dimension. The fore-body shape consists of R_N = 1.35 m, a half-angle cone of R_N = 50 deg, D = 3.4 m, L = 1.508 m, and R_C = 0.01 m. The OREX geometry incorporates a rear cover with a small backward facing step at the junction between back cover and heat shield. The aft body is having α_B = 15 deg, half-angle cone relative to the plane of symmetry.

There is no doubt that the aerothermodynamics of the APOLLO capsule are one of the best known. Due to the large number of flights in the 1960s and 1970s either in Earth orbit or of Lunar return, the free-flight data base is remarkable [4]. The OREX free-flight experiment gives the possibility to study how intensive is the influence of physical phenomena along the re-entry trajectory [5].

IV. BOUNDARY CONDITIONS

A. Initial Conditions

The free-flight experiment gives the possibility to study how intensive is the influence of physical phenomena along the re-entry trajectory on the wall heat flux [5]. In [6], viscous shock layer (VSL) and Navier–Stokes (NS) solutions were generated for selected trajectory points with various degrees of modelization, where at the wall, a temperature distribution was prescribed. The general trend is that for less than 100 km altitudes the computations with slip conditions give the more realistic answer [7].

The important factor affecting convergence of the two problems is the desired boundary conditions. Both the capsules were examined in this paper occurred during free stream conditions[10] at 76 km and 85 km flight conditions are given in Table I the subscript ∞ represents free-stream value.

Table I: Initial boundary conditions

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

B. Modeling Turbulence

Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities.

The k-ε turbulence model is a two-equation model in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The model has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. For the reentry flight simulations, the axisymmetric turbulent predictions with the *k*–ε models are in reasonable agreement with the flight measurements [11]. The k-ε model is used in the present paper to validate the turbulent flow over reentry bodies.

C. Grid Sensitivity

One possibility for the poor agreement between the computation and experiment is grid resolution. In order to examine this effect, grid refinement studies were conducted covering a range of mach numbers 3-5 based on geometry. The number of grid points- in the interval count was varied to determine their effect on computed flow.

Present work has done with three levels of grids in order to check the grid dependency over re-entry bodies. One grid was meshed with an edge interval count of 100, second was with 150 and a third (fine mesh) with an edge interval count of 200. Results are shown only for 200 interval count mesh. From the study made on three levels of meshes only afterbody region is varied a little in the range of 10^{-2} compared to the fine mesh. Forebody flow properties varied less than 1.3% on all grids tested (slightly larger deviations were seen at the separation point due to numerical issues with some grids of interval count 100 and 150).

The baseline grid has 200 points along the body surface and 120 points in the normal direction. Wall spacing was chosen to maintain a constant cell Reynolds number, which implies that the near-wall spacing in the low-density afterbody is much larger than on the forebody.

V. RESULTS AND ANALYSIS

During the development and flights of APOLLO, no numerical methods with adequate quality for complete flow field simulations at corresponding trajectory points were available. Pre-flight predictions and post-flight comparisons were done with analytical relations mainly developed for predicting the forward stagnation point heating. This paper demonstrates the importance of understanding the effects of shock waves and illustrates how small change in flight attitude can alter the resulting aerodynamic forces on the capsule. Finally, any minor alteration to the shape of the craft will also have great implications. The temperature fields along the two vehicle bodies, OREX and FIRE II, will be discussed.

The process of diminishing the error was done iteratively using the CFD tool Fluent. After a number of the cases were reviewed in which the solver gave the acceptable results for the fine grid for which the solutions are presented. Flow simulations were carried at different Angles of Attack (AoA), with initial freestream conditions listed in Table I. The solutions are in the range Mach 3 to 5, which spans the shock variations along its entry into atmosphere.

D. Physical Interpretation

The computed velocity vector plots over FIRE II & OREX are shown in Fig. 3. The red lines indicate areas of higher intensity, while the blue just the opposite and the circulation and behind the body there is a disk shock. Close-up views of the velocity vector plot over fore-body of the capsules and schematic shock location are depicted in the figure.

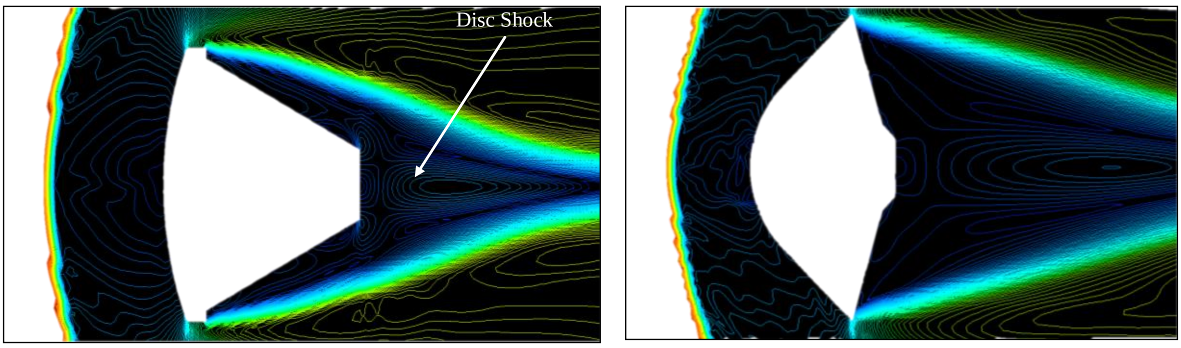

Fig. 3 Shock induced flow over FIRE II & OREX at hypersonic speeds

E. Comparison between Supersonic and Hypersonic flows

Comparision of flows is shown in the fig. 4 and 5, It is observed that the distance between the shock wave and the heat shield of the OREX capsule decreases from Mach 3 to 5 i.e., the shock wave formed comes closer to the body with increase in mach number.

Comparing with theoretical normal shock relations formula, i.e., affable

$$M_2^2 = \frac{1 + \left[\frac{(\gamma - 1)}{2}\right] M_1^2}{\gamma M_1^2 - (\gamma - 1)/2}$$

where the M_1 is the mach number before shock and M_2 is after the shock and γ we get the following results;
$M_2 = 0.474$ for supersonic speed (Mach 3)
$M_2 = 0.415$ for hypersonic speed (Mach 5)
The computational values of Mach number across the shock wave are;
$M_2 = 0.481$ for supersonic speed (Mach 3)
$M_2 = 0.417$ for hypersonic speed (Mach 5)
As observed, the theoretically calculated values and the values obtained from FLUENT contours for the mach number immediately after the shock wave are matching. Similar trends were observed for FIRE II re-entry capsule.

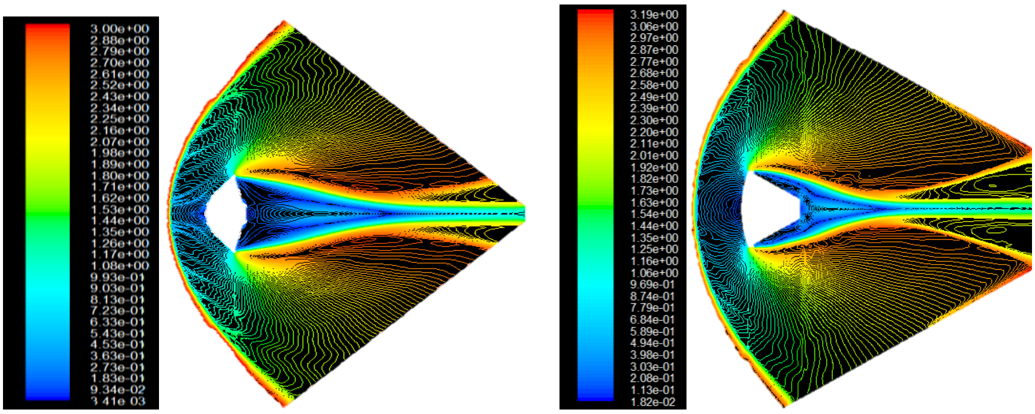

Fig. 4 Mach contour lines at Supersonic Speed over OREX & FIRE II.

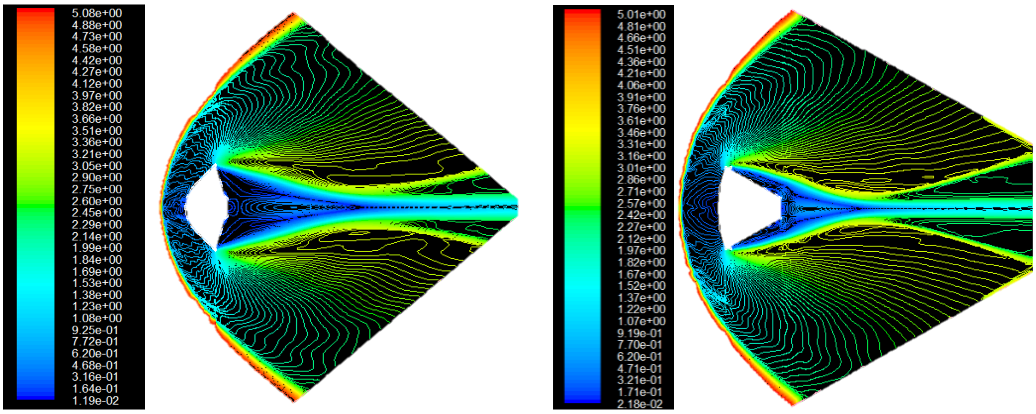

Fig. 5 Mach Contour lines at hypersonic speeds over OREX & FIRE II.

F. Effects of Angle of Attack

The angle-of-attack has a definite effect on the physical properties of the craft. In Fig. 7 the vehicle is at an angle of -20°, while in Fig. 7 at an angle of -40° and at -60° in Fig. 8. Results such as these are important to understand in the sudden case that the re-entry vehicle becomes unstable and begins to rotate freely, in hopes of understanding a way to counteract this occurrence. At -40 AoA (fig.7), illustrating the mach contour, we can see the unique structures forming in the tail portion of the flow, as flow comes close to rest on the heat shield of the vehicle, as well as on the rear. Changing the angle even further will illustrate further the great shift in dynamics. Fig. 7 displays the mach contour lines that would result with an angle of -40°, while Fig. 8, with an angle of -60°, shows the velocity contour lines that would result. Fig. 8 clearly shows how this change in angle affects the flow behind the re-entry vehicle. If the FIRE II capsule was coming at a zero degree angle, we could expect the streamlines just off the rear of the spacecraft to be also at a zero degree angle. However, with an angle of -40°, we observe how the streamlines would cross, causing a turbulent flow. With fig. 9, this is the first instance we find the shock switch from a normal shockwave to an oblique shockwave. This is because the normal shock wave changes to oblique when the angle, formed between the surface of the vehicle and the free stream, exceeds 45°, as one can see with this angle-of-attack of -60°. The normal shockwave over the heat shield is desired as this change in kinetic energy is what slows the re-entry vehicle as it enters the earth's atmosphere at such great speeds as a Mach number of 2.5.

We will not observe this same change in energy with an oblique shockwave; therefore it is vital to keep this angle-of-attack from shifting less than 45° in either direction.

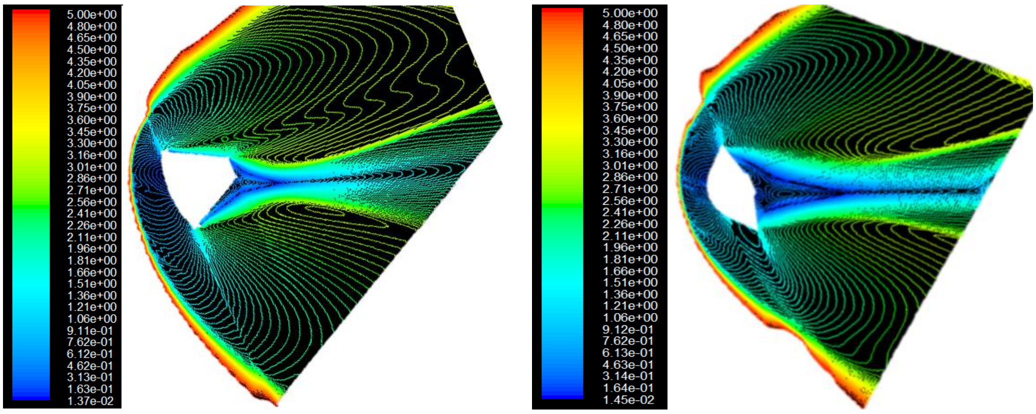

Fig. 6 Mach Contour lines over FIRE II at -20 AoA

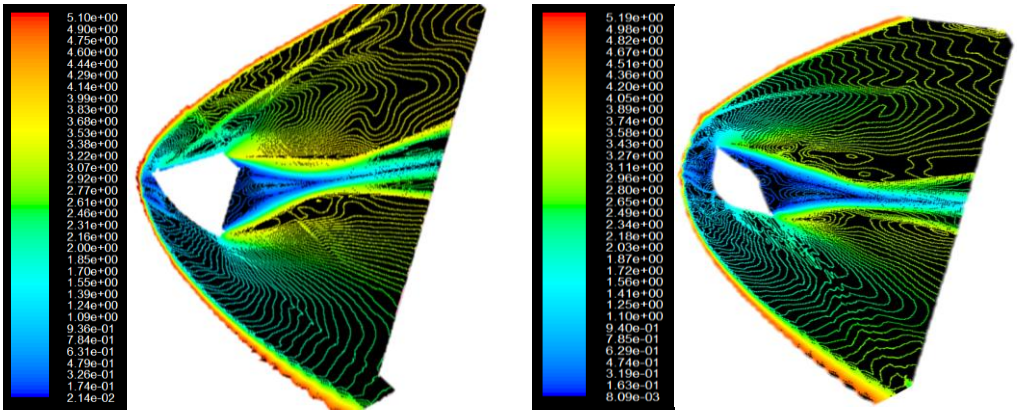

Fig. 7 Mach Contour lines over FIRE II & OREX at -40 AoA

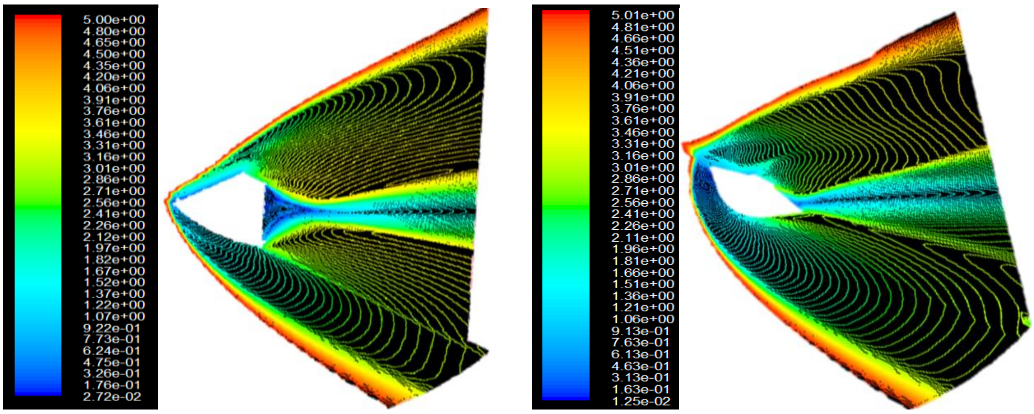

Fig. 8 Mach Contour lines over FIRE II & OREX at -60 AoA

G. Vehicle Shape Analysis

For different shapes of re-entry capsules, there is a long tail that extends out from the back, whose shape will differ between re-entry vehicles. To illustrate the difference between the FIRE II and OREX designs, we may observe the temperature variances as seen in Fig. 9. Here we can see that the general shape of the shock wave itself is practically identical, however, the differences lie in the rear of the object. The most dangerous difference is the great amount of heat generated on the rear of the craft.

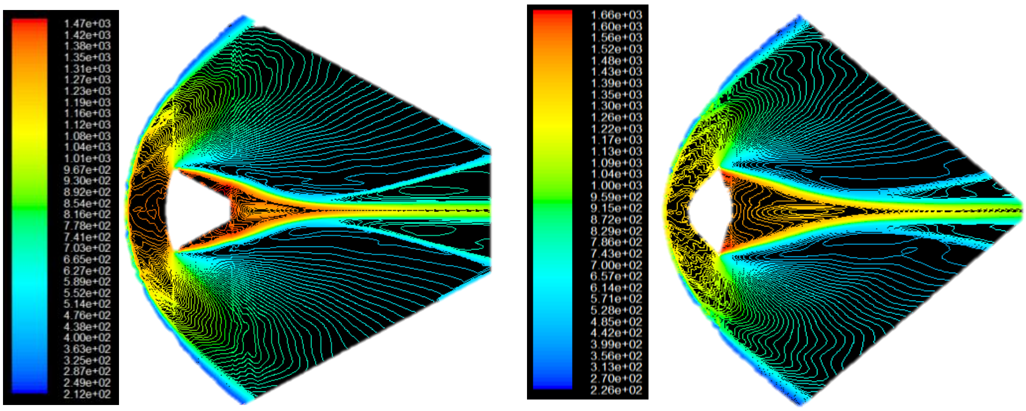

Fig. 9 Temperature Contour lines over FIRE II & OREX at hypersonic speeds

VI. CONCLUSIONS

The Apollo shape re-entry capsule is one topic that has resurfaced as a result of dependability. Due to this interest, the analysis of such a craft especially with the increasingly better technological methods such as the advancements in CFD, has also become very important. This paper exemplifies this importance. Physically, the aerodynamic forces effect on the motion of re-entry vehicle, especially at such high speeds, that even a slight change in angle of attack can severely alter the activity of the re-entry capsule including the shock way which plays such a big factor in the fate of the craft. A slight change in the vehicle shape can alter the dynamics of the craft. As the vehicle reaches lower altitudes a lift maneuver acts on the vehicle and creates a large impact footprint. It does qualitative analyses of the patterns or rules associated with aerodynamic forces as they follow changes in the geometrical parameters of vehicles or bodies, having important reference value for the design of reentry bodies or vehicles. Characteristics flow features around the blunt body at supersonic & hypersonic speeds is observed. The high surface pressure on the fore-body results the development of high aerodynamic drag which is required for the aerobraking application. Results adequately explain that, in a situation where angle of attack changes**,** the asymmetries in the nose section cause flow fields to be non-uniform, and, in conjunction with that, there are obvious influences on the afterbody.

REFERENCES

[1]. R C Mehta, *Computations of flow field over Apollo and OREX reentry modules at high speed*, Indian Journal of Engineering & Materials Sciences, , December 2008, Vol.15.

[2]. Michael Wright, Mark Loom and Periklis Papadopoulos, *Aerothermal Analysis of the Project Fire II Afterbody Flow*, AIAA, 2001.

[3]. R C Mehta, *Numerical Simulation of Supersonic Flow Past Reentry Capsules*, Journal of Shock waves, Vol. 15, 2006.

[4]. Ernst-Heinrich Hirschel, Claus Weiland, *Aerothermodynamic Design Problems of Hypersonic Flight Vehicles*, Springer, 2008.

[5]. Hirschel, E.H., *Basics of Aerothermodynamics*, progress in Astronautics and Aeronautics, AIAA, Reston, Vol.204. Springer, 2004.

[6]. Roop N. Gupta, Jim J. Jones, and William C. Rochelle, *Stagnation-Point Heat-Transfer Rate Predictions at Aeroassist Flight Conditions*, NASA TP-3208, NASA-Langley, 1992.

[7]. *M. S. Ivanov and S. F. Gimelshein, Computational Hypersonic Rarefied Flows, Annual Rev. Fluid mech. Vol. 30:pp. 469–505, 1998.*

[8]. Braun, R. D. and Manning, R. M., *Mars Exploration Mars Exploration Entry, Descent and Landing Challenges*, Paper 1076, 2006 IEEE Aerospace Conference, Big Sky, Montana, March 2006.

[9]. Mitcheltree, R. A., DiFulvio, M., Horvath, T.J., and Braun, R. D., *Aerothermal Heating Predictions for Mars Microprobe*, AIAA Paper No. 98-0170, Jan. 1998.

[10]. R C Mehta, *Computations of Flowfield over Reentry Modules at High Speed*, Computational Simulations and Applications, 2009.

[11]. Christopher J. Roy and Frederick G. Blottner, *Assessment of One- and Two-Equation Turbulence Models for Hypersonic Transitional Flows*, Journal Of Spacecraft and Rockets, Vol. 38, No. 5, September–October 2001.

Contributors

Mr. Shiva Prasad U. obtained his M. Tech (Aerospace Engineering) from Institute of Aeronautical Engineering, Hyderabad, A.P. Presently, he is working as Asst. Prof. in the Dept. of Aeronautical Engineering, Manipal Institute of Technology, Manipal University (MU-MIT), Manipal. His research interests include: CFD, Hypersonic Aerodynamics, Propulsion and Combustion. He has Four International conference publications to his credit.

Mr. Srinivas. G obtained his M. Tech (Aerospace Engineering) from Institute of Aeronautical Engineering, Hyderabad, A.P. Presently, he is working as Asst. Prof. in the Dept. of Aeronautical Engineering, Manipal Institute of Technology, Manipal University (MU-MIT), Manipal. His research interests include: CFD, Aerodynamics, and propulsion. He has 4 conference publications to his credit.