Flight Analysis of Wedge and Blunt Nose Re-entry Capsules

Vinu R.G.[#1], Kailas Sree Chandran[#1], Deepak G[#1]

[#1]Assistant Professor, Department of Mechanical Engineering, MCET
Kerala, India

Abstract—*Re-entry vehicles returning from space are exposed to very high velocities in atmosphere. The air friction along can cause very high temperatures to be developed at the vehicle wall. So the geometrical of the vehicle is of prime interest in designing the vehicle. The experiment on this project concentrates on understanding the various characteristics of hypersonic flow and re-entry vehicle and the reliability on CFD software for determining the flow characteristics. The analysis compares the flow characteristics using a wedge based and blunt nose based models. The capsule is modelled using Design Modeller and meshed in Ansys. The analysis is carried out using CFX. K-omega 2 equation SST model is used for turbulent modelling in CFX analysis. K-epsilon model is also used for comparing the results. Profiles of pressure, temperature and density are mainly studied.*

Keywords—*Blunt nose, flow characteristics, hypersonic flow, re-entry capsule.*

I. Introduction

The practical design of hypersonic vehicles for sustained hypersonic flight in the atmosphere will be a major challenge to the next generation of aerospace engineers. The designing and modernization of a re-entry space vehicle requires accurate and reliable data on the flow field, aerodynamic characteristics, heat transfer processes. Taking into account the wide range of flow conditions, realized at hypersonic flight of the vehicle in the atmosphere, it leads to the need of incorporating various features occurring in the gas phase and on the vehicle surface. Analysis of flow characteristics of re-entry vehicle is of interest for engineers over a decade. Re-entering into the earth atmosphere occurs as a free fall and has a velocity in hypersonic range. Since the analysis using experimental setup is difficult to achieve, engineers has to relay on computational fluid dynamic software for predicting the flow characteristics before running any trials. In the present work such an analysis is carried out with the aim of determining various hypersonic characteristics of different geometries of re-entry capsule. The selected capsules for this study are wedge inspired FIRE II model and blunt nose type CARE module. The capsule is modelled using Design Modeler, meshing and analysis is carried out using CFX. Boundary layer developments and comparisons, thermal distribution over the domain, analysing the effect of angle of attack etc are obtained from the analysis..

II. Experimental Procedure

A. Modeling and Meshing

The models chosen for wedge type nose analysis is The Flight Investigation of Reentry Environment (FIRE) II. The model chosen for blunt nose type re-entry capsule is the test flight model of ISRO CARE. The module name CARE is an abbreviation of Crew Module Atmospheric Re-entry Experiment .The modeling of the modules is carried out from the dimensions available from flight data.

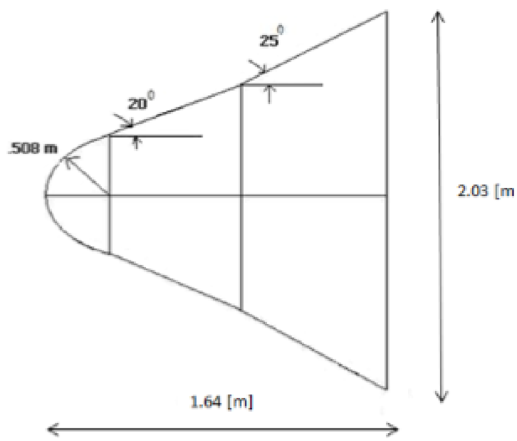

Fig 1. Dimensions of Fire II

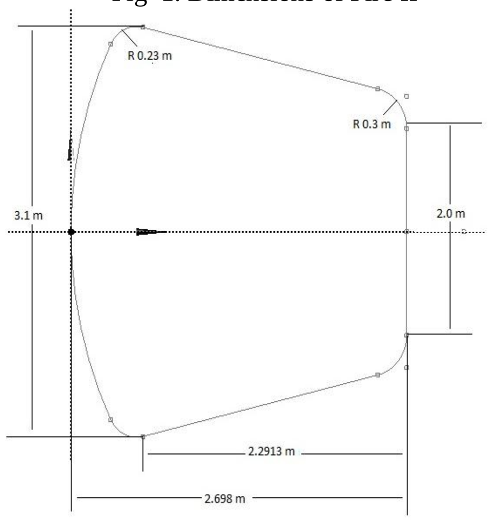

Fig 2. Dimensions of CARE module

After modeling the capsule the flow field is created. The inside of the flow field is defined as a fluid and walls as solid. The model used for analyzing the flight parameters are made at an angle of attack of 0 degree.

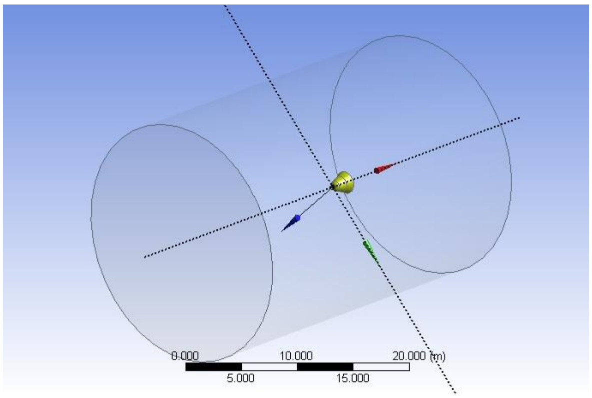

Fig 3. Flow field and capsule

Meshing is carried out using CFX mesh. The entire flow field is meshed as fine elements. The capsule wall is meshed using finer elements and the corners of the capsule wall are meshed more fine. Edge and facing sizing's are used for achieving this. This reduces the computation time along with providing accurate results as the boundary layer development is close near the walls

.

B. Analysis

The analysis is carried out in Ansys 15 CFX software. The meshed model is imported to CFX and default domain parameters are set. Air is considered as an ideal gas for the analysis. The fluid model chosen for heat transfer is Total energy based. Viscous work term is also incorporated for including the effects of viscous interaction to the flow. Turbulence model chosen is Shear stress transport model with automatic wall function.

Since the flow in hypersonic regime is compressible, high speed wall heat transfer is also enabled in the analysis. The high temperature generated by the hypersonic flow due to friction, dissociation of oxygen and nitrogen molecules may occur. But for the present study the effect of chemical reactions is not considered. So no chemical or combustion reactions are enabled for in setting up the analysis.

The boundaries chosen for the study are the inlet of the flow field, outlet and the wall of the CARE module. The operating pressure was set to 0 Pa, to decrease the chance of numerical error due to the low pressures resulting from the solution [1]. The temperature of the capsule wall is chosen as isothermal. The boundary conditions for the analysis are taken from the flight data for the altitude. Fluid in flow field is air and considered as an ideal gas for the calculation. Temperature of the inflow air is 265 K and no species reactions are considered. The analysis is carried out for different Mach no from 7.5 to 22.5 in the hypersonic velocities. The highest Mach no for re-entry vehicle ranges around 25.

III. RESULTS AND DISCUSSIONS

A. Pressure

The total pressure value is found maximum at the blunt nose end with a value of 1.89×10^7 Pascal and varying to 7.36×10^6 Pascal near the curved wall and a minimum of 1.25×10^6 at the tail end. The high pressure at the nose region is a result of molecules imparting pressure. It shows the severe pressure drag at the two edges of the module base. High static pressure is created in the base of the reentry vehicle as illustrated in since, the pressure is high while re-entering in to the atmosphere due to the strong bow shock created. This bow shock will increase drag force acting on the re-entry vehicle and has the capability to decelerate the vehicle to low Mach numbers. The maximum static pressure is created at the far field of the re-entry vehicle because of the progressing bow shocks marching downstream of the vehicle.

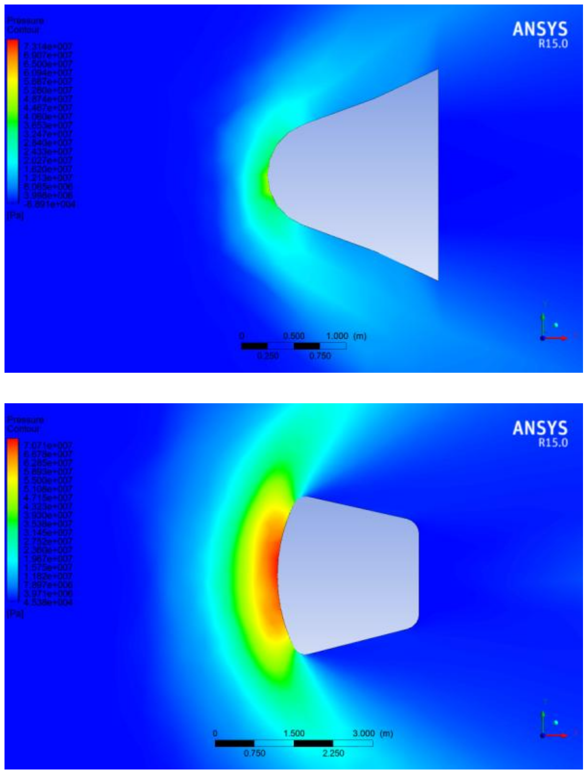

Fig 4. Pressure profile at Mach 22.5 for FIRE II and CARE capsules

The difference in pressure at nose of module is measured for both models and at different Mach no.

The plot for the variation of the pressure values at different Mach no for both models is shown in fig 5

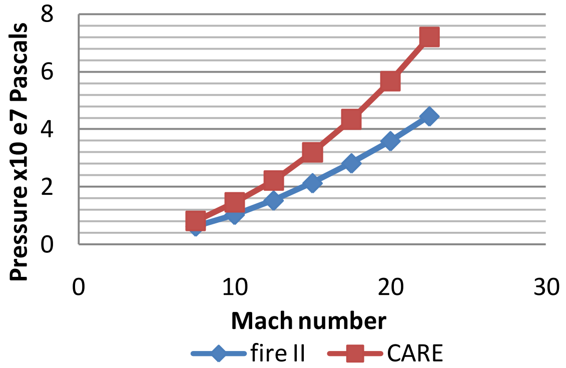

Fig 5. Variation of pressure with Mach no

From the graph it is observed that at low Mach numbers the maximum pressure produced is nearly same. The pressure value at Mach 7.5 for FIRE II is 0.628 x 10^7 Pa and that fore CARE module is 0.814 x 10^7 Pa. SO there is still a difference of 2 x 10^6 Pa for both models. As the velocity increases the difference in maximum pressure value increases. For Mach 22.5 the maximum pressure observed in FIRE II module is 4.45 x 10^7 Pa whereas for CARE module it peaks to 7.2 x 10^7Pa. This very high pressure value for the blunt nose design makes it difficult for the vehicle to sustain 0^0 Angle of impact at high velocities.

B. Temperature

The temperature distribution around the blunt nose at hypersonic velocities is a result of frictional flow of air molecules around the flight. The maximum is observed at the front end since the flow is maximum there. Fig shows the simulation of the temperature contours over the capsule. Here we can see, the temperature is maximum at the heat shield and it is also observed that the potential as well as kinetic energy decreases. So according to the law of conservation, if some energy function decreases, some other energy should be increasing. Here the kinetic and potential energy is decreasing and it is dissipating in the form of heat energy. The maximum temperature value obtained is 12439.3 K. The maximum temperature is produced at the nose of the re-entry vehicle and it is lowest amount at the sloping edges. Minimum value is 5301.64 K.

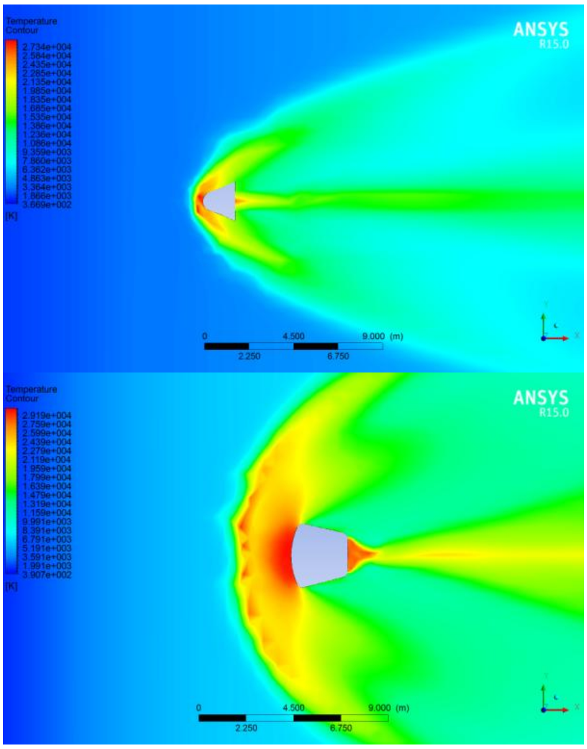

Fig 6. Temperature profile at Mach 22.5 for FIRE II and CARE capsules

The maximum temperature value for a reentry vehicle is observed at the nose of the capsule. This is because of the conversion of high velocity of air into heat. This is the conversion of kinetic energy of air into heat energy at walls due to friction. The blunt nose has a larger area of impact than the wedge model. So the high temperature is distributed along the entire wall. This causes an increase in the heat shield area which is to be provided at the nose. Ceramic tiles and heat shields are to be used in front of the aluminum alloy panels for withstanding this high amount of temperature.

The maximum temperature values at the nose for both designs at different Mach no is measured and plotted as below.

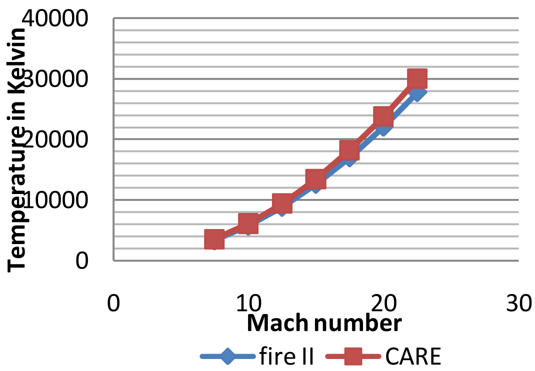

Fig 7. Comparison of maximum temperature for FIRE II and CARE

From the analysis it is observed that the maximum temperature value for both models is almost same. The differences increase with increasing velocity. The nose temperature for FIRE II at Mach 22.5 is 27757.5 K and for CARE is 29985.6 K. The difference in temperature at very high velocity is around 2000 K. For a blunt nose design this high temperature exists along a large area increasing the need for insulation.

C. Density

The density contour gives an idea about the changes in the gas phase occurring around the flight. The profile at a Mach no of 22.5 is shown as below.

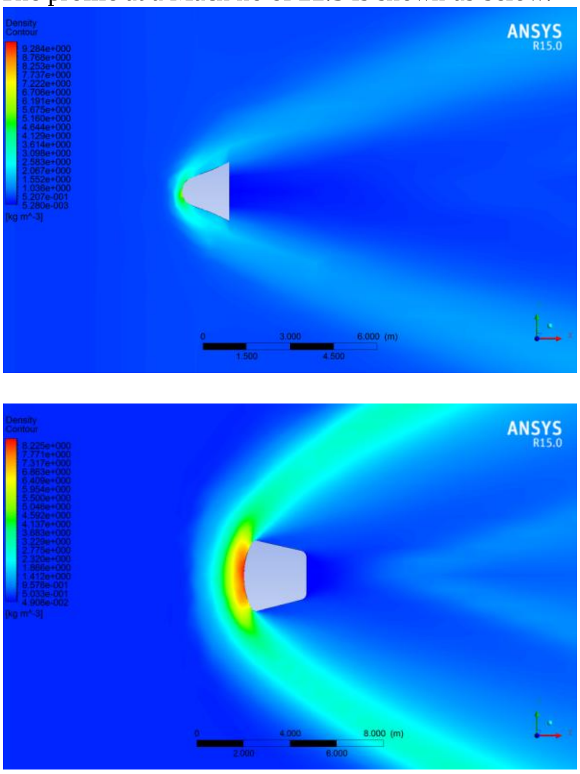

Fig 8. Density profile at Mach 22.5 for FIRE II and CARE capsules

The shock wave created by the vehicle resisting high velocity air flow causes a change in the density of air around it. The density of air is observed maximum at the nose potion as the flowing air is resisted by the nose causing a compressing of the air. If the density of air is not symmetrically distributed it will affect the flight stability. For both designs it is observed that the density change is distributed symmetrically around the vehicle. The large nose area of the blunt nose increases the amount of high density air at nose. The geometrical shapes of the CARE module make the air to be distributed to a large area producing a very distinct shock wave. Also the air is directed far from the vehicle without causing it to slide through the sides of the vehicle.

The variation of density at nose for wedge model and blunt model is shown measured and plotted to compare the results. This is shown in fig

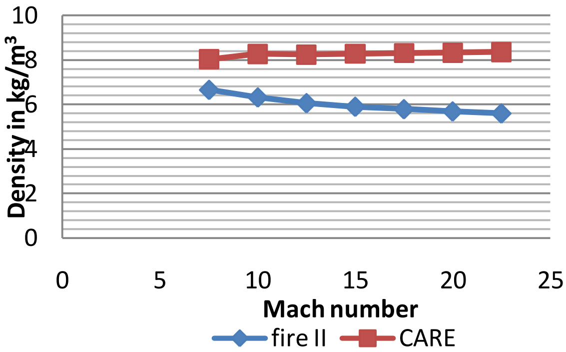

Fig 11. Variation of Density with Mach number

It is observed that the density value remains almost constant for both model on its on specific values. The density for FIRE II model is ranging between 6.66 kg/m^3 to 5.6 kg/m^3 for a change in Mach no from 7.5 to 22.5. The density for CARE model is ranging from 8.04 kg/m^3 to 8.37kg/m^3. It is also observed that the density value shows an increase with increase in velocity for blunt nose model whereas a decrease for FIRE II model. This is associated with the shock wave created by geometry of the vehicle.

IV. CONCLUSIONS

The CFD analysis is effective utilized for carrying out the experiment. The geometry of re-entry capsule plays an important role in the distributing of flow parameters around the vehicle. It is observed that the blunt nose design can be effectively used for future re-entry missions. The design increases the amount of heat insulation but this can be overcome by the advantage of increased safety. Safety will be increasedby the fact that the conversion of kinetic energy into heat reduces the impact damage considerably when coming down to earth.

The design of blunt nose shows increase in maximum temperature and density at the nose of the vehicle. But this increase is not so high than a wedge based model. The small differences in these parameters can be overcome by suitable selection of material.

The study has proved the effectiveness of using commercial CFD software for predicting the results in a hypersonic flow condition. The different aspects of a flight such as pressure shock wave profile, temperature distribution etc are generated. More computational aerodynamic tools are required for the successful aerodynamic design and analysis of trajectory vehicles. Wind tunnel tests are important in the validation of prediction methods if they are not available, the aerodynamic analyst should consider the use of multiple independent codes to test the results for consistency.

REFERENCES

[1] Dr.B.Balakrishna, S. Venkateswarlu and Dr P. RavinderReddy "*Flow Analysis of an Atmosphere Reentry Vehicle*" International Journal of Engineering Research and Development Volume 3, Issue 4 (August 2012), PP. 52.

[2] Dr. Roy N Mathews A, Shafeeque A "*Hypersonic Flow Analysis On An Atmospheric Re-Entry Module*" , International Journal of Engineering Research and General Science Volume 3, Issue 5, September-October, 2015 ISSN 2091-2730

[3] Louis M.G.Walpot, Michael J.Wright, Peter Noeding, FerrySchrijer, "*Base flow investigation of the Apollo AS-202 Command Module*", Progress in Aerospace Sciences 48–49 (2012) 57–74.

[4] Shiva Prasad U and Srinivas, "*Flow Simulation over Re-Entry Bodies at Supersonic & Hypersonic Speeds*" International Journal of Engineering Research and Development , Volume 2, Issue 4 (July 2012), PP. 29-34

[5] 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.

[6] Krishnendu Sinha, " *Computational Fluid Dynamics in Hypersonic Aerothermodynamics*" Defence Science Journal ,Vol. 60, No. 6, November 2010, pp. 663-671

[7] Y. Matsudaa,H. Kiharab, K. Abeb , "*Numerical Study of Thermochemical Nonequilibrium Flow around Reentry Capsule and Estimation of Aerodynamic Heating*", 7th Asian-Pacific Conference on Aerospace Technology and Science, 7th APCATS 2013

[8] V.Carandente, R.Savino1, M.Iacovazzo and C.Boffa, "*Aerothermal Analysis of a Sample-Return Reentry Capsule*", FDMP, vol.9, no.4, pp.461-484, 2013 Tech Science Press

[9] Bruce Ralphin Rose. J, Saranya. P, "*High Temperature Flow Characteristics over a Re-Entry Space Vehicle*", International Journal of Latest Trends in Engineering and Technology (IJLTET).

[10] P Ramya, A Hemanth Kumar, JaswanthMoturi, N.Ramanaiah"*Analysis of Flow over Passenger Cars using Computational Fluid Dynamics*", International Journal of Engineering Trends and Technology (IJETT), V29(4),170-176 November 2015

[11] F Menter, "*Two-Equation Eddy-Viscosity Transport Turbulence Model for Engineering Applications*", AIAA Journal 32(8), sept 1994 pp 1598-1605

[12] Bruce Ralphin Rose. J,Saranya. P, "*High Temperature Flow Characteristics over a Re-Entry Space Vehicle*", International Journal of Latest Trends in Engineering and Technology (IJLTET),Vol 4, Issue 1, May 2014

[13] AnoopThankachen, Santosh kumar"Design Optimization and Analysis of Rocket Structure for Aerospace Applications", International Journal of Engineering Trends and Technology (IJETT), V24(6),286-291 June 2015

[14] Sandeep J, Dr B Balu, A. G. Sarwade "*Review paper on Hypersonic Air Intakes*" International Journal of Innovations in Engineering and Technology, Volume 6, Issue 3, February 2016

[15] Manuel D. Salas, "*A Review of Hypersonics Aerodynamics, Aerothermodynamics and Plasmadynamics Activities within NASA's Fundamental Aeronautics Program*",AIAA 2007-4264

[16] T.Karthikeyan, S.K.Aravindhkumar, J.Arun Kumar"*Design and Analysis of Aerospike Nozzle to Improve Thrust in Hybrid Rocket Engine*", International Journal of Engineering Trends and Technology (IJETT), V36(7),347-351 June 2016