Hypersonic Flow Analysis of CARE Capsule

Vinu R G[1], Raj Ganesh R[2]

PG Scholar, JCM CSI IT, Trivandrum, Kerala, India [1]

Assistant Professor, JCM CSI IT, Trivandrum. Kerala, India [2]

Abstract: 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. 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 selected model for study is ISRO CARE test flight. 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 velocity are mainly studied. The effect of angle of attack on the flight profile is also studied.

Keywords: CARE, flow characteristics, hypersonic flow, re-entry capsule.

I. INTRODUCTION

Air breathing hypersonic flight is held by many to be the last frontier of air-vehicle design. Some progress has been made, but much need to be done. 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. 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 ISRO CARE re-entry capsule. The capsule is modeled using Design Modeler, meshing and analysis is carried out using CFX. Boundary layer developments and comparisons, thermal distribution over the domain, analyzing the effect of angle of attack etc are obtained from the analysis.

II. PAGE LAYOUT

Several works have been carried out in the field of implementing computational fluid dynamics for analysing hypersonic flow characteristics. In the present work mainly 4 works has been taken for understanding in carrying out the experiment.

Computational fluid-dynamics results can be used in evaluation of the aero thermodynamic analysis of re-entry trajectory and to show the flow field around a blunted cone-flare in hypersonic flow [1]. The studies are usually carried out using 2D models because of the difficulties in computation. However 3D model analysis was carried put to obtain the flow field that develops around reentry capsules [2]. CFD software and analyses can also be used

to determine the effects at different mach numbers and different angles of attack [3]. The importance of the aerodynamic forces effect on the motion of re-entry vehicle, especially at such high speeds is 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 [4]. 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.

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 [5]. The heat flux distribution on the capsule surface and to perform one-dimensional thermal analyses for its ablative heat shield can be studied from CFD analysis [6]. Physical modeling and numerical methodology-related issues involved in hypersonic flow simulation are of consideration in analyzing using CFD software [7]. Numerical simulation of the flow fields of Apollo AS-202 shows influence of the laminar-turbulent transition position in the boundary layer around a reentry capsule on the heat flux at the wall.

III. EXPERIMENTAL PROCEDURE

A. Modeling and Meshing

The model chosen for analysis is the test flight model of ISRO. The module name CARE is an abbreviation of Crew Module Atmospheric Re-entry Experiment .The modelling of the CARE module is carried out from the dimensions available from ISRO news reports. After modelling the capsule the flow field is created. The inside of the flow field is defined as a fluid and walls as solid.

The modelling is done for different angles of attack. This include minute variations of 4^0 and 6^0 and major rotation of 10^0 20^0 and 40^0. The model used for describing the results here is made at an angle of attack of 0 degree.
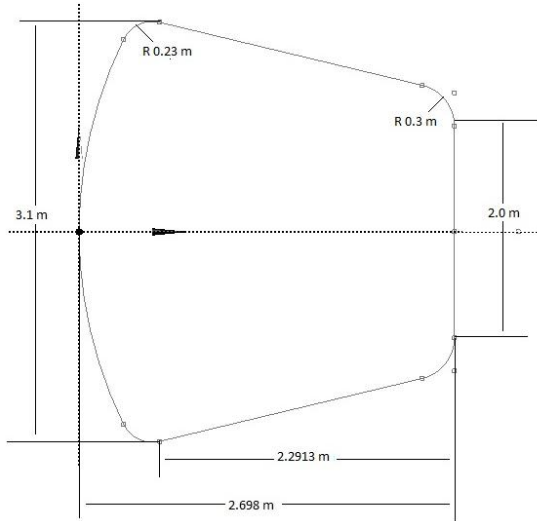

Fig 1 Dimensions CA RE module

For meshing in CFX a fine sized elements are chosen with advanced edge sizing on the different regions around the CARE wall body. This is accomplished by the use of spitting the domain into different regions and specifying edge meshing on the individual regions. For this we can use the option edge sizing, face sizing and inflation. By this method the computation efficiency can be maintained along as a relative coarse meshing is used in the flow domain far from the CARE wall.
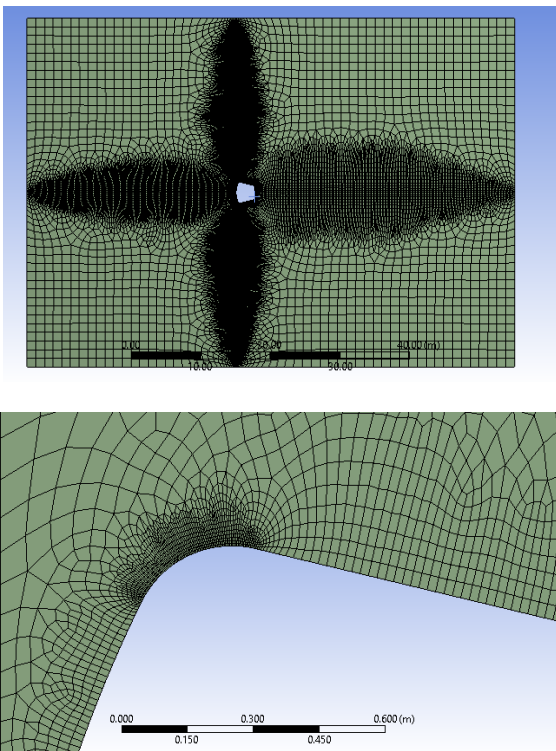

Fig 2. Meshing of Flow Field and CARE wall

Also many accurate results can be produced as the meshing near to the CARE wall is made fine and again finer meshing on the corners of the wall.

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 CARE wall is chosen as isothermal. The boundary conditions for the analysis are taken from the ISRO CARE flight data for the altitude 73.3 km and Mach no 16.67. 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.

IV. RESULTS AND DISCUSSIONS

A. Profiles

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.

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.
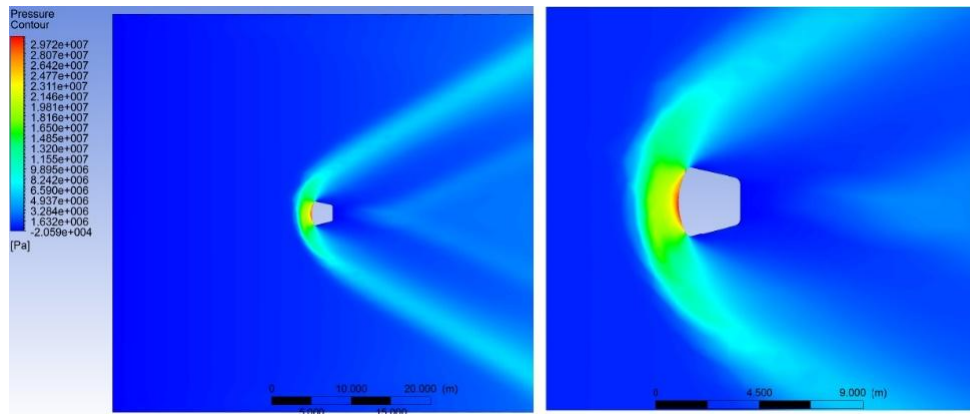
Fig 3. Pressure profile at Mach 16.6

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.

The velocity contour gives an idea about the corners about which the flow velocity will be maximum. The profile at a velocity of 5 km/s is shown as below.
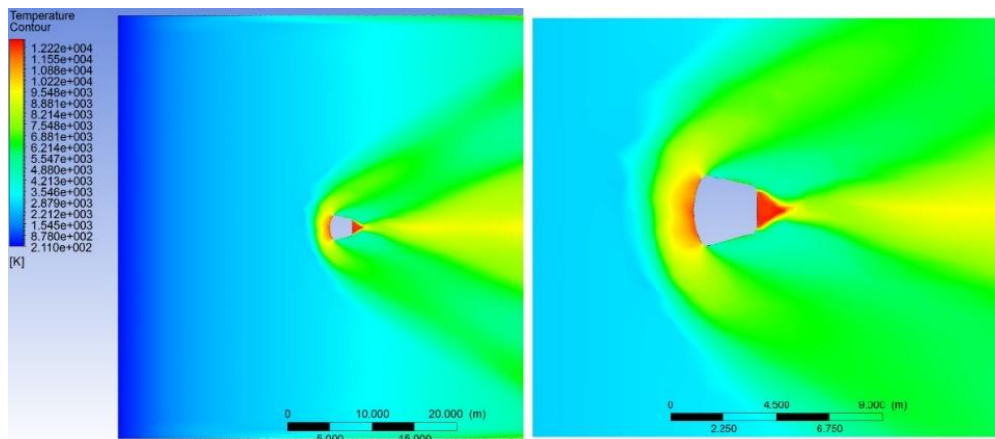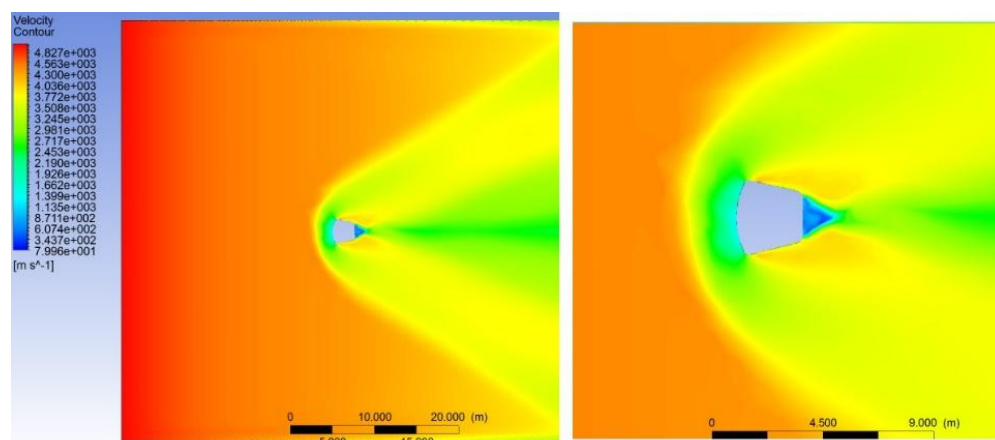

Fig 4. Temperature profile at Mach 16.6

Fig 5. Velocity profile at Mach 16.6

The shock wave created by the CARE wall in the flow field will cause the velocity of the domain also to form distinct separate regions. The red portion indicated the maximum velocities, ie is the free stream velocities. The flow is obstructed by the CARE wall and forming a minimum value at the nose of the module.

This is represented by the blue portion. Also the flow is directed along the corners of the wall forming a stream line which converges back at the tail portion. The region behind the module is obstructed by the solid wall in front so a minimum velocity is observed here also. From the experiment results it is observed that when considering the velocity near to the solid wall it is maximum at the corners.

B. Temperature variation around CARE module

The temperature variation around the CARE module is symmetrical along the axis of the module by virtue of the shape of the module. This is observed by analyzing using a 3D model for analysis. The distribution of temperature around the module is as shown below
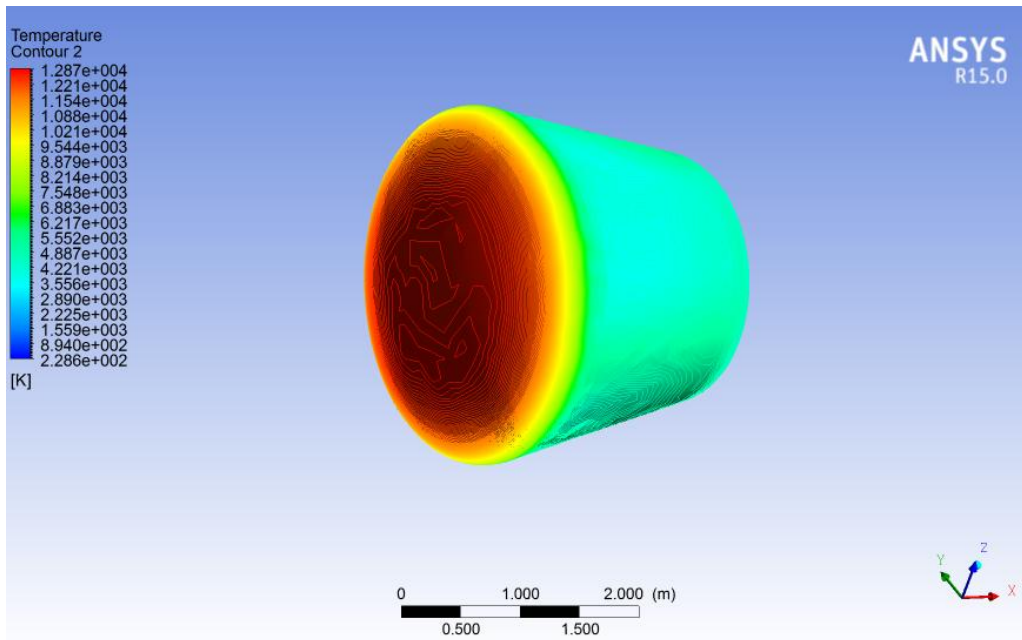

Fig 6. Temperature profile around CARE wall

The variation of the temperature around the front diameter of the CARE module is shown in fig 7
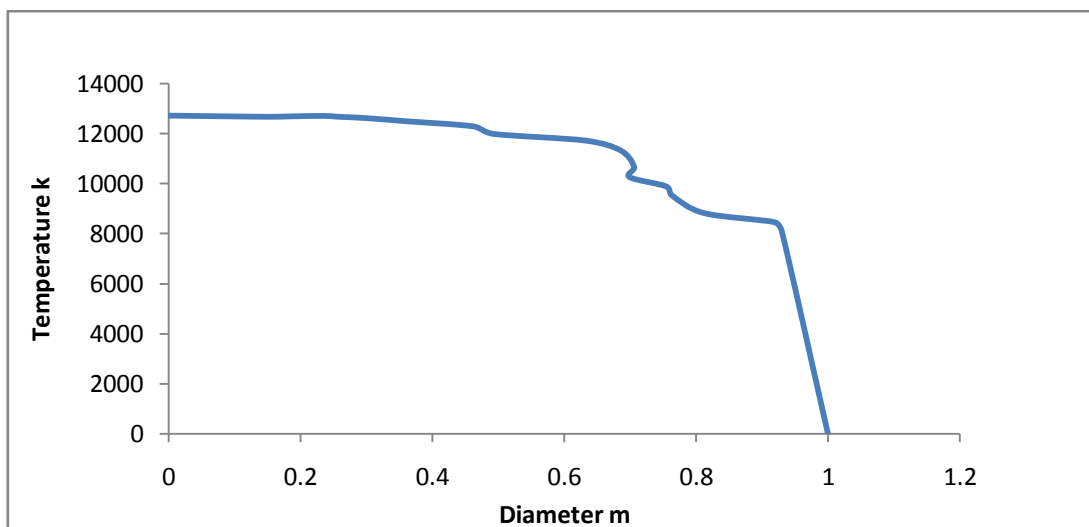

Fig 7 Variation of temperature with diameter of CARE

From the graph it can be understood that the temperature around the care wall reduces abruptly when the diameter becomes maximum. This is because of the geometrical shape of the wall which obstructs the free stream velocity in a nearly perpendicular direction. So a large portion of the wall is directly affected by the high temperature produced by conversion of kinetic energy to thermal energy.

The variation of temperature along the boundary layer formed in front of the nose is fig 8

It is seen that there is a sudden drop in temperature after a distance of 2 m in front of the capsule and remain almost constant after a distance of 3 meters.
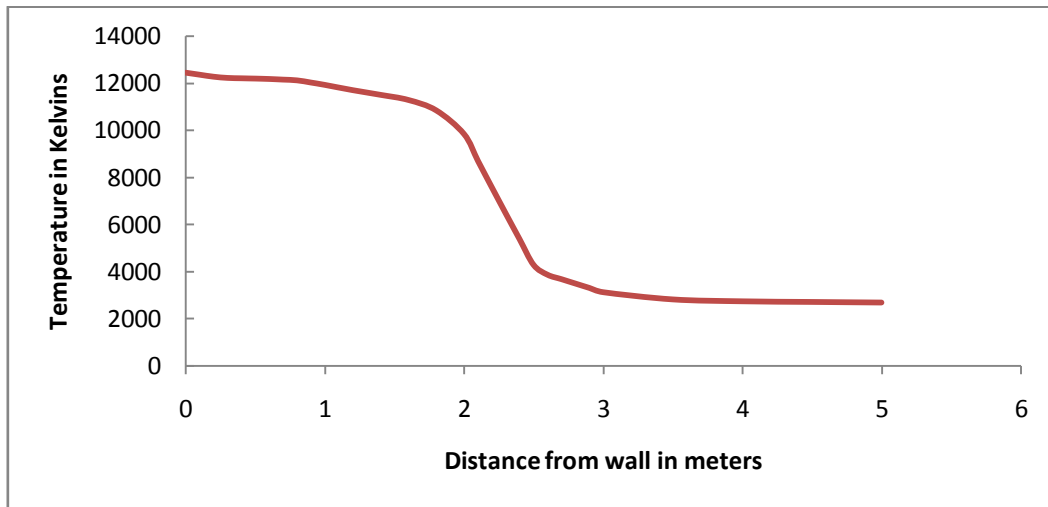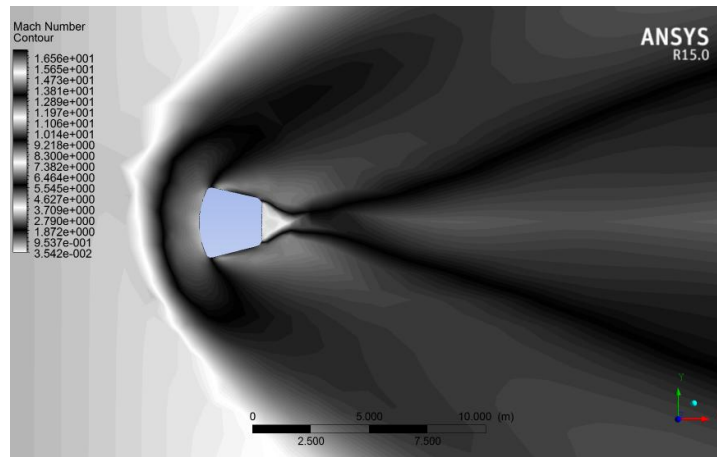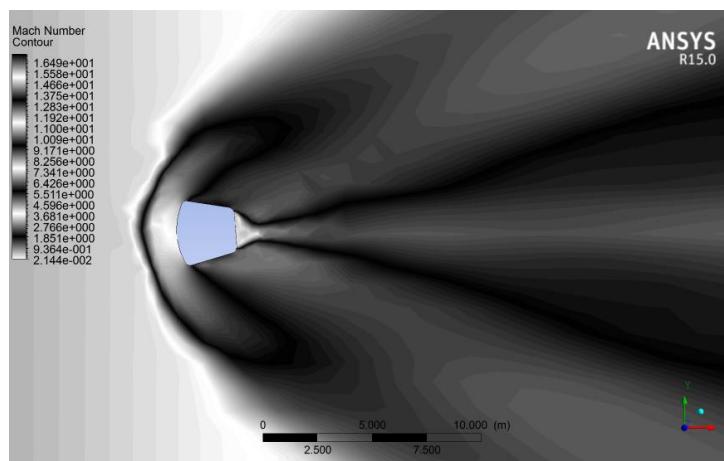

Fig 8. Variation of temperature along boundary
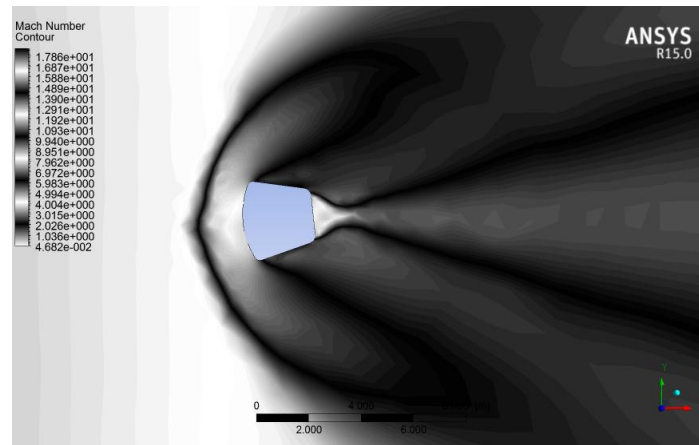
C. Effect of angle of attack

The effect of change in minor angle of attack in the mach profiles around the re-entry capsule for different angles is as shown
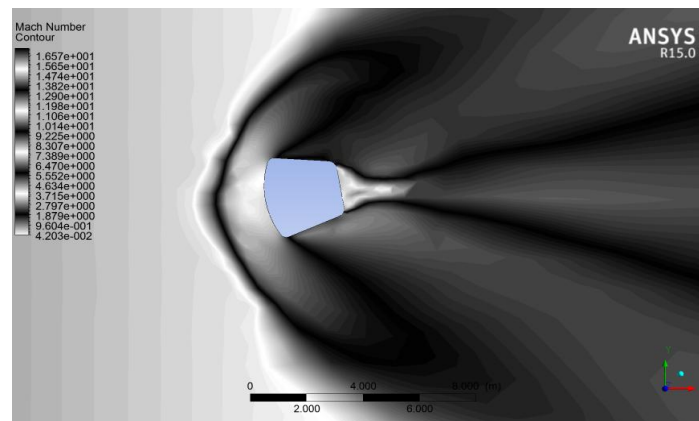

Mach no profile at AOA 0^0
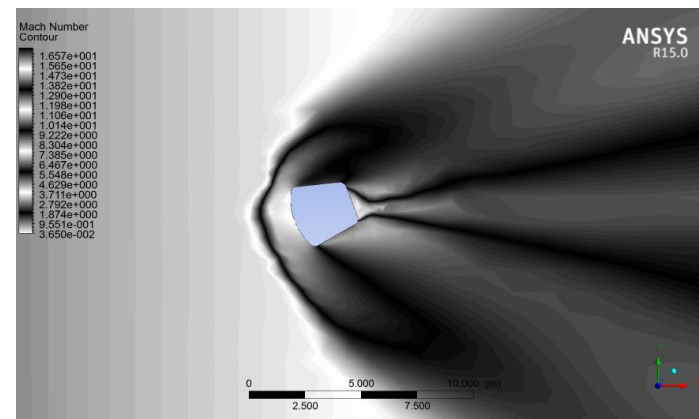

Mach no profile at AOA 4^0

Mach no profile at AOA 6^0

Fig 9. Effect of small angle of attack

For a wedge shaped model the angle of attack affects the flow stability to serious extents. But from the analysis for the new design of capsule it shows that the small changes in angle don't affect the flight much. Even if there occur a change in impact angle the flight can regain stability. But the maximum temperature values at the front end will be moved to the corners slightly. As the angle of attack increases the effects will be more predominant and should be considered.

The angle-of-attack has a considerable effect on the physical properties of the craft especially at higher angles.

Results such as these are important to understand in the sudden case that the re-entry vehicle becomes unstable and begins to rotate freely. At 60^0 AoA, 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. If the CARE 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.

Mach no profile at AOA 10^0

Mach no profile at AOA 20^0
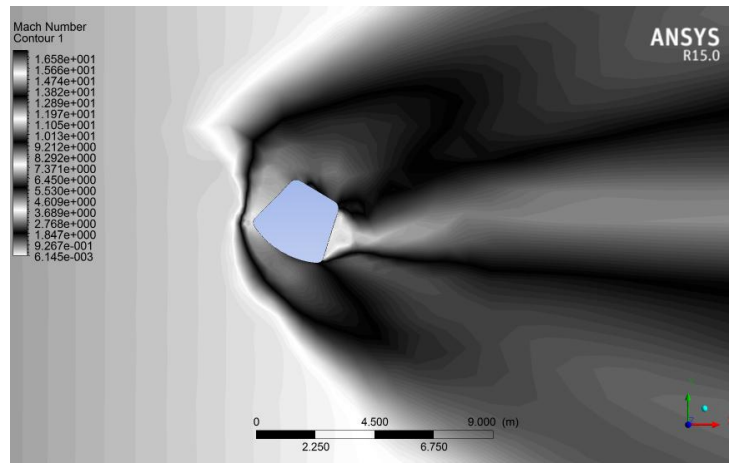
Mach no profile at AOA 60^0
Fig 10. Effect of Major angle of attack

However, with an angle of 60°, we observe how the streamlines would cross, causing a turbulent flow. 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 0.8.

V. CONCLUSION

Applied CFD has been used extensively during the design and analysis of the re-entry module. CFD methods can be used routinely for the prediction of aerodynamic characteristics of unusual and unconditional flight vehicles. It is an essential fact which is learned through the aerodynamic design and analysis using advanced simulation process. But we have to depend upon different types of analysing methods for obtaining accurate results regarding various features. From this investigation, it is found that this method can offer aerodynamic information on a timely basis while keeping the cost and schedule of commercial programs.

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. Also the effect of rotation of the vehicle in flight is also produced from the study.

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. Ravinder Reddy "Flow Analysis of an Atmosphere Reentry Vehicle" International Journal of Engineering Research and Development Volume 3, Issue 4 (August 2012), PP. 52-57

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

[11] Raj Ganesh, "Thermal analysis of a Re-entry vehicle"2010

[12] Ananthu, "Thermal Analysis of the Wings of a Re-Entry Vehicle"2011