International Journal of Emerging Technologies in Engineering Research (IJETER)
Volume 5, Issue 4, April (2017) www.ijeter.everscience.org

Coupled Analysis of Re-Entry Vehicle

L.Jayaprakash
Assistant Professor, Department of Mechanical Engineering, Shivani College of Engineering &Technology, Tiruchirappalli, Tamilnadu, India.

P.Viswabharathy
Assistant Professor, Department of Mechanical Engineering, Shivani College of Engineering &Technology, Tiruchirappalli, Tamilnadu, India.

G.Athiram
UG Scholar, Department of Mechanical Engineering, Shivani College of Engineering &Technology, Tiruchirappalli, Tamilnadu, India.

P.Nishanth
UG Scholar, Department of Mechanical Engineering, Shivani College of Engineering &Technology, Tiruchirappalli, Tamilnadu, India.

Abstract – The main goal of the work described in this paper was to setup a procedure for modeling a thermal production system for hypersonic reentry vehicle by the author. A multiphysics framework has been setup for the simulation of hypersonic reentry vehicle using commercial codes CFD and FEA with user defined programming CFD (FLUENT**) and the material thermal and structural response code (ANSYS) are loosely coupled to achieve the solution. The evaluation of the aero thermodynamics analysis of reentry trajectory. CFD results are presented to show the flow field around a capsule in hypersonic flow. FEA results are presented to show the stress around a capsule in hypersonic flow. CFD analysis represents a key technology within planetary entry vehicle design. Safe landing of vehicles re-entering from space requires, in fact, an accurate understanding of all physical phenomena that take place in the flow field past the hypersonic vehicle to assess its aerodynamics and aerothermodynamics performance. These parameters are of primary relevance for the design of reentry trajectory and vehicle thermal protection system; for the latter, in particular, it is pointed out to role catayticity on the vehicle thermal load. In this framework, a possible earth-entry scenario for the proposed capsule type vehicle is reported and analyzed.**

Index Terms – Reentry Vehicle, CFD Analysis, (FLUENT), FEA.

1. INTRODUCTION

Space capsule will survive re-entry through the Earth's atmosphere or that of another planet and impact on the surface requires knowledge of the forces of gravity and acceleration along with test design trials. Many early spacecraft that orbited the Earth landed on land or water which is still quite a hard surface if you are traveling at high speed. The Mercury, Gemini and Apollo spacecraft all landed in the water with the aid of parachutes. The Russian Soyuz spacecraft landed and still lands on land with the aid of parachutes and jet firings. The Mars Pathfinder crash landed on the surface of Mars in 1997 with the aid of parachutes and protected by airbags.

Blunt body configurations are the most common geometries employed for entry into planetary atmospheres. The two of the biggest external forces that a space capsule experience are gravity and drag. Drag is the space capsule’s resistance to it being pushed though air. Air is a mixture of different molecules, including nitrogen, oxygen and carbon dioxide. Anything falling through air hits these molecules and therefore slows down.

The amount of drag on a capsule depends on many things, including the density of the air, and the shape, mass, diameter and roughness of the capsule. The speed of a space craft highly depends on the combined effect of the two forces gravity, which can speed up a rocket, and drag, which will slow down the rocket. Space capsules entering Earth’s atmosphere will be considerably slowed because our atmosphere is so thick.

1.1 AERODYNAMIC HEATING

The heating induced by the very high speeds of reentry of greater than Mach 20 is sufficient to destroy the structure of the vehicle. The early space capsules such as those on Mercury, Gemini, and Apollo were given blunt shapes to produce a standoff bow shock. As a result most of the heat is dissipated to surrounding air without transferring through the vehicle structure. Additionally, these vehicles had ablative material that sublimates into a gas at high temperature. The act of sublimation absorbs the thermal energy from the aerodynamic heating and erodes the material away as opposed to heating the capsule. The Space Shuttle uses insulating tiles on its lower surface to absorb and radiate heat while preventing conduction to the aluminum airframe. The compromise of the heat shield during liftoff of Space Shuttle Columbia contributed to its destruction upon reentry. During reentry the vehicle suddenly heats due to the dissipation, in the boundary layer, of its high energy (potential and kinetic) by friction with the atmosphere. Knowing the free stream density, flight speed, nose and wing leading edge radii, and enthalpy variation with temperature, the stagnation point heat flux can be computed, thus providing a preliminary assessment of the vehicle aero-heating environment.

1.2 BALLISTIC REENTRY

The body or the probe falls on the surface of earth through the atmosphere under the influence of drag and gravity falling at point *a* with impact fig1.3. the vehicle produce a very little amount of lift or it produce no lift and lands or falls on a predetermined location according to the condition at which it is entering initially into the atmosphere, since there is negotiable lift force is produced therefore the opposing force is in the direction of flight of the body. The pilot has no control over his landing position during this type of reentry. All entries except of a space shuttle are ballistic entry, and it is also our project concern as most of the part or reentry of hybrid space probe Capsule will be covered by ballistic reentry. The most important parameter used in controlling the trajectory of the flight during re-entry is the Ballistic Coefficient. For a low value of, the deceleration as well as the heating is less intense than for a high value of as we know that the entry occurs at a very high atmosphere having less air density. The Inter-Continental Ballistic Missiles (ICBM) initially utilized this reentry method with high blunted sphere-cone-cylinder-flare geometries. Later it was discovered that the efficiency and accuracy could be enhanced by increasing the ballistic coefficient value by using a blunted sphere cone geometry thereby increasing its impact velocity such that there is a very less effect of wind on the descent phase. Thermal protection is provided thereby allowing the material to melt and vaporize and thereby transferring most of the heat back into the atmosphere. This method is called "ablation," and the material used in vehicle's surface is known as an "ablator". We will find out the effect of on different parameters related to the atmospheric reentry by using computational methods which are fast and easy to calculate after finding the relation of with respect to other parameters such as velocity, deceleration, Mach number and Reynolds number.

1.3 SOLUTION METHODOLOGY

The steps of setting up a problem in FLUENT are discussed briefly: defining geometry, importing and checking the grid, selection of solver formulation and equations to be solved (laminar/turbulent/inviscid etc.), material properties, specification of operating and boundary conditions, specification of numerical properties (under-relaxation factors, CFL etc.) and initialization of variables. The criterion for convergence was in the order of 10-3 for continuity, x, y and z velocities and 10-5 for energy calculation and turbulence quantities k and ω .The residuals were monitored in the graphics window of FLUENT. In addition the net mass flow rate was monitored for convergence. For all the cases iterations continued till the convergence or near convergence were reached.

2. LITERATURE REVIEW

[1] Estimation & CFD analysis of Re-entry Parameters for Hybrid Space

Probe Capsule

With the help of FORTRAN we calculated the safe and different parameters for reentry, entry velocity for lifting body is calculated according to mass of the body, and different bodies have to enter at different velocity land safely to the ground. On comparing four different configurations we found Ballistic – lifting configuration suitable for entering to the earth's atmosphere among lifting, ballistics and lifting ballistic configuration. The results are much safer than the other three. The body is touching the ground at the velocity of 10.56m/s with suitable maximum deceleration unlike other configurations. We obtain suitable impact velocity but not the desirable maximum deceleration values. After simulating the flow we conclude that most of the heat dissipation is over during initial phase of atmospheric reentry. And this work also concluded that at higher altitude the Knudsen number less than 0.01 so the flow doesn't behave like a fluid behaving at high Knudsen number i.e. in atmosphere. Direct Simulation Monte Carlo method is proposed for the simulation of the flow at higher altitude. Which solve the Boltzmann equation which takes account of intermolecular distance in the flow molecules. It solves the equation with the help of Jacobian matrices.

[2]Aerothermodynamics of blunt body entry vehicles

The aero thermodynamic phenomena of blunt body entry vehicles are discussed. Four topics will be considered that present challenges to current computational modeling techniques for blunt body environments: turbulent flow, non-equilibrium flow, rarefied flow, and radiation transport. Examples of comparisons between computational tools to ground and flight-test data will be presented in order to illustrate the challenges existing in the numerical modeling of each of these phenomena and to provide test cases for evaluation of computational fluid dynamics (CFD) code predictions.

Detailed information on each of the test cases presented herein is available and it is recommended that each be studied further in detail and used in the assessment of modern computational tools. However, these data sets themselves have significant uncertain- ties and are not inclusive enough of all physical situations to be considered adequate to fully-validate numerical tools used in the design of an actual reentry vehicle. At best, these data can help to provide conservative upper bounds on the uncertainties of pre-dictive methods. It is therefore recommended that both ground testing and flight testing of blunt body aero thermodynamic phenomena, coupled to computational predictions and analysis, be rigorously pursed, and that aero thermodynamic instrumentation be included as an integral part of all future missions.

[3]Computational Analysis Of Effect Of Spiked Reentry Capsules On Reduction In Temperature

Experiments and computations were performed to understand the flow field around a scaled down model of Apollo re-entry capsule at Mach 6. Temperature distribution measurements were made for various aero spikes added to the model. Computations using the commercially available software FLUENT, were carried out for two dimensional axisymmetric flow. Three different types of aero spikes, namely: arrow head, aerospike, aerodisk were used in front of the re-entry capsule to increase the distance between the body and the shock wave and hence decreasing the temperature. The effects of the three different types of aero spikes on the flow with various dimensional variations are analysed as well. From the analysis, it was found that the aerodisk yield a rise in reattachment temperature at the shoulder of the Apollo capsule.

The basic flow structure around the re-entry capsule was successfully captured through computational analysis. Using spikes as well as aerodisks we can alleviate the high levels of aeroheating on the blunt body. Even though the aerodisk helps in reducing the aeroheating level on the blunt body, it degrades the efficiency of the spike itself. i.e, the pointed spikes are much efficient and advantageous over aerodisks. The aerodisk and arrowdisk with longer length ratios provide much more efficient cooling overall. But an increase in length for the spike counter-act the cooling due to the intensity of the re-attachment shock created. The benefits of the lowered heating of the capsule come at a price of a very high heating of a small area, the spike.

[4]Aerothermal Analysis of a Sample Return reentry Capsule.

The aerothermal analysis of a sample-return hypersonic capsule reentering on Earth from an interplanetary exploration mission. The main objective of the work is to estimate the heat flux distribution on the capsule surface and to perform one-dimensional thermal analyses for its ablative heat shield. the numerical models implemented are described and the computational results, obtained along a feasible reentry trajectory, are presented and discussed. Particular attention has been paid to compare the convective stagnation point heat fluxes obtained by means of Computational Fluid Dynamic (CFD) analyses with the ones computed with engineering correlations. A further comparison between CFD and with Direct Simulation Monte Carlo (DSMC), in order to investigate the air rarefaction effects, is reported.

The main purpose of the present work was to calculate aerothermal loads occurring during an Earth reentry of a sample-return capsule from super orbital speeds. CFD analyses have been performed in four points of the computed reentry trajectory, assuming a possible baseline capsule configuration, both in non-catalytic and in fully-catalytic conditions. Due to the large amount of energy characterizing the atmospheric reentry, dissociation and ionization processes are so developed that the fully-catalytic heat flux at the stagnation point at the altitude of 60 km is about twice the corresponding non-catalytic value. Comparing CFD results at the investigated altitudes with three different semi-empirical models a reasonably agreement with the Detra-Hidalgo correlation was observed. a one-dimensional thermal analysis for the capsule ablative heat shield has been carried out at the stagnation point, in order to preliminary dimension the material thickness necessary to maintain the bondline temperature below a maximum tolerable value.

[5]Flow analysis of an atmosphere reentry vehicle.

Computational fluid-dynamics results are presented to show the flow field around a blunted cone-flare in hypersonic flow. This is of particularly interest since it features the hypersonic flow around planetary reentry vehicles, the reason between the cone and the flare is particularly critical with respect to the evaluation of the surface heat flux. Indeed, flow separation is induced by the shock wave boundary layer interaction, with subsequent flow re attachment, that can dramatically enhance the surface heat transfer. The exact determination of the extension of the recirculation zone is particularly delicate task for numerical codes. A numerical approach has been adopted to study the flow field that develops around the EXPERTS capsule.

At high Mach number of vehicle, velocity of flow increased and the pressure of the vehicle decreases drastically. At low Mach number of vehicle, there is an effect on vehicle skin will be corrodes by high temperature due to low Mach number of vehicle used. The reentry vehicle behaves differently at different Mach numbers. At Mach numbers 5 and7 the temperatures on the vehicle increases from 2460 0K and 7840 0K respectively.

3. CAD MODEL

3.1 SPACE SHUTTLE CAPSULE

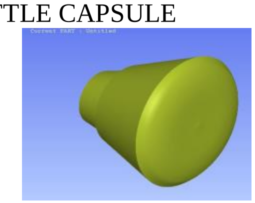

Figure 3.1 height=3.8 m width= 5 m length=5 m

3.2 SPACE SHUTTLE CAPSULE WITH WIND TUNNEL

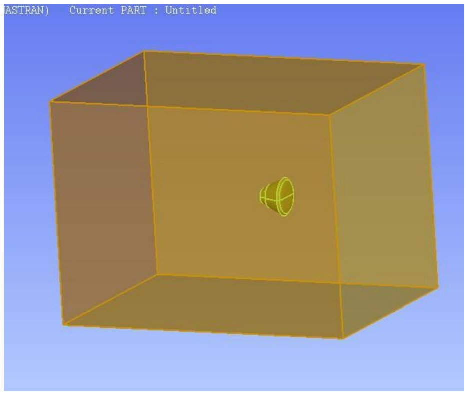

Figure 3.2 wind tunnel construction front= 4L back= 5L side=2L where, L= height of capsule

3.3 SURFACE MESH

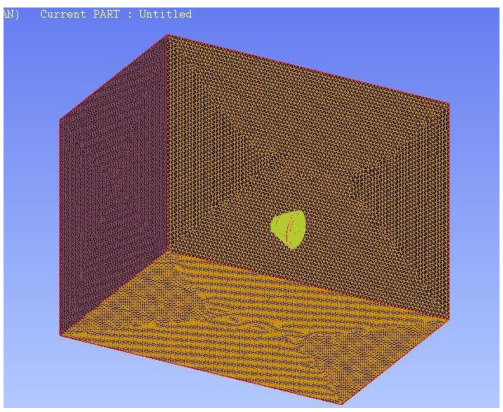

Figure 3.3Surface mesh count = 66102

Skewness = 0.6

3.4 VOLUME MESH

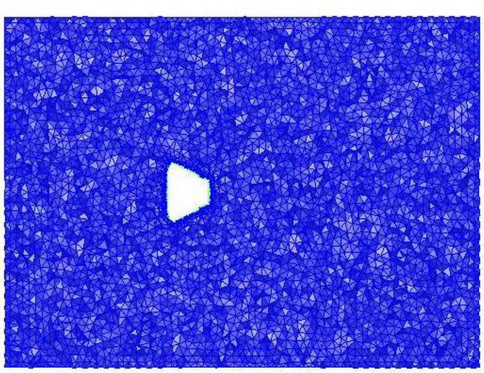

Figure 3.4 Volume mesh count = 792432 cells

Skewness = 0.84

4. FLUENT CONDITIONS

Table 4.1 Solver Settings

Processing	Serial
Solver	coupled, Pressure Based
Dimension	3D

Table 4.2 Problem Setup

Materials	Type		Properties	Unit
	Fluid	Air	Density -1.225 Cp-1006.43 k-0.0242 Viscosity-1.78e-5	Kg/M3 J/Kg k W/Mk Kg/Ms
	Solid	titanium	Density -2719 Cp-871 k-202.4	Kg/M3 J/Kg k W/Mk
Turbulence Modeling			Spallart-allamars	
Energy Equation			On	
Cell Zone Condition			Fluid-Air	

Table 4.3 Boundary Conditions

	Fluid Flow		Thermal
Inlet	Pressure far field	Mach no=0.75	Temperature-245.5 K
Outlet			
Wind tunnel wall			
Capsule	Wall	No slip condition	

Table 4.4 Equation Solved

Equation solved	Flow	Conservation of mass
		Conservation of momentum
	Energy	Conservation of energy
	Turbulence	Turbulence modelling-spallart allmaras.

Table 4.5 Software Used

Pre-Processing	Ansa	Domain Extraction Geometry Cleanup Surface Meshing Surface Mesh Cleanup
	Ansys-Tgrid	Volume Meshing
Solver	Ansys-Fluent	Fluid Flow And Thermal Behaviour
Post-Processing	Ansys-Fluent	End results

5. RESULTS

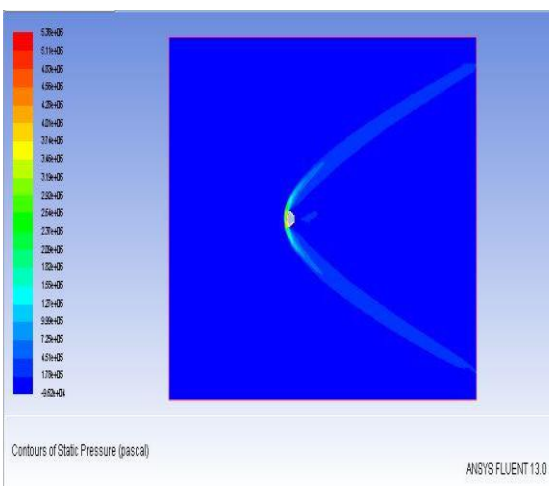

Figure 5.1contours of static pressure around the capsule along the surface.

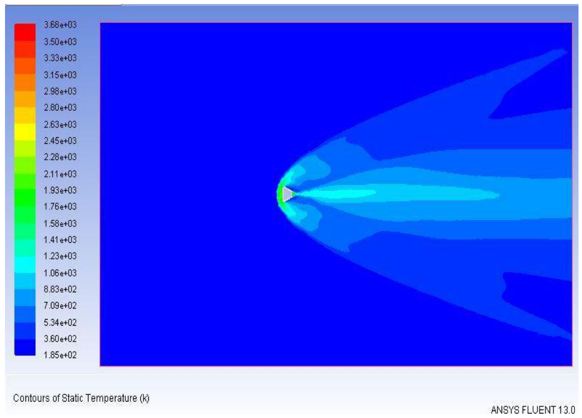

Figure 5.2 contours of static temperature around the capsule along the surface

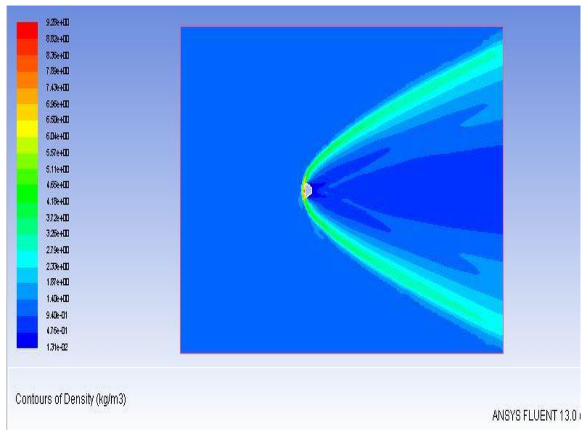

Figure 5.3contours of density around the capsule along the surface

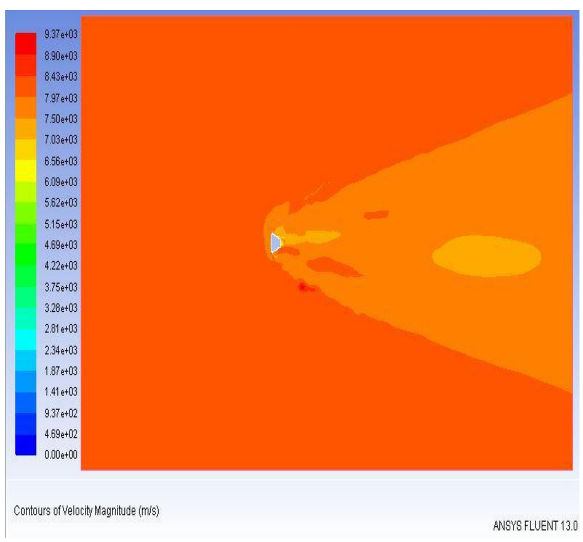

Figure 5.4 contours of velocity magnitude around the capsule along the surface

6. CONCLUSION

Thus, the space re -entry shuttle capsule is analyzed for the Mach number (Ma=25). Thus, the post-processing results of the flow field around the capsule when capsule attains Ma=25 is predicted. when flow is above Ma=0.3, the flow nature changes to compressible flow, so, the fluid experiences pressure change around the capsule results in shock waves and also due to compressibility , the fluid density changes. Due to high pressure and viscous force, it experiences the high temperature distribution to the fluid and also to the capsule, due to tangential shear forces of fluid.

REFERENCES

[1] Burkhardt, J., Zimmerman, F and Schottle. U. M., ―Operational use of Guided Reentry Capsules – Systems Design Solutions and Mission Safety Considerations‖, Journal of Aerospace Science and Technology. Vol.8, Issue 7, pp 635-644, Oct 2004.

[2] Costa, R. R., Silva J. A., Wu, S.F., Chu, Q. P and Mulder J.A., ―Atmospheric Reentry Modeling and Simulation‖, Journal of Spacecraft and Rockets, Vol. 39, No. 4, pp 636-639, Feb 2002.

[3] Desai, P. N., Mitcheltree, R. A and Mcneil Cheatwood, F., ―Sample Returns Missions in the Coming Decade‖, IAF-00-Q.2.04, 2000.

[4] FLUENT Incorporated, ―Fluent 6.1.2 User's Guide‖ CD-ROM documentation with FLUENT Version 6.2 Software, April 2005.

[5] Fujimoto, K and Fujii, K., ―Computational Aerodynamic Analysis of Capsule Configurations toward the Development of Reusable Rockets‖, Journal of Spacecraft and Rockets, Vol. 43, No. 1, pp. 77-83, Jan-Feb 2006.

[6] M.YM.Ahmed, N.Qin, ―Numerical Investigation of Aeroheating Characteristics of Spiked Blunt Bodies at Mach 6 Flight Conditions‖, The Aeronautical Journal, June 2011, Volume 115, No 1168.

[7] Anderson, J. D. (1989): Hypersonic and High Temperature Gas Dynamics. McGraw-Hill.

[8] Bird, G. (2004): The ds2v/3v program suite for dsmc calculations. In Capitelli, M.(Ed): Proceedings of the 24th International Symposium on Rarefied Gas Dynamics,Monopoli, Italy.

[9] Aerothermal Analysis of a Sample-Return Reentry Capsule 481

[10] Boyd, I.; Trumble, K.; Wright, M. (2010): Modeling of stardust entry at high altitude, part1: Flowfield analysis, nasa ames research center. Journal of Spacecraft and Rockets, vol. 47, no. 5.