International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2013): 6.14 | Impact Factor (2013): 4.438

Aero-Thermodynamic Analysis of Expert Capuslereentry Vehicle Using CFD

Narendranath Alokam[1], B. Dinesh Kumar[2], D.Muppala[3]

[1]PG Scholar (Cad-Cam), MLRITM Dundigal, Hyderabad, Telangana, India

[2]PG Scholar (Aerospace), MLRITM Dundigal, Hyderabad, Telangana, India

[3]Associate Professor MLRITM Dundigal, Hyderabad, Telangana, India

Abstract: ***The main task of the project is the evaluation of the Aero-thermodynamic analysis of reentry trajectory. Computational fluid-dynamics results are presented to show the flow field around a blunted cone–flare in hypersonic flow. This problem is of particular interest since it features most of the aspects of the hypersonic flow around planetary entry vehicles. The region 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 reattachment, that can dramatically enhance the surface heat transfer. The exact determination of the extension of the recirculation zone is a particularly delicate task for numerical codes. A numerical approach has been adopted to study the flow field that develops around the EXPERTS capsule. Laminar flow computations have been carried out using a full Navier–Stokes solver, with free stream condition with two different Mach number. A simplified dissociative model based on the Monti-Napolitano method has been implemented into the commercial CFD code FLUENT. The CFD solution is abating to study the 3D flow filed around the EXPERTS capsule. The results and description performed as for the results obtained by the numerical solution.***

Keywords: CFD, Fluent, Nose cone, Reentry trajectory, Mach number.

1. Introduction to the Atmospheric Reentry

Atmospheric reentry is the movement of human-made or natural objects as they enter the atmosphere of a planet from outer space, in the case of Earth from an altitude above the Karman Line, (100 km). The *Karman line* lies at an altitude of 100 km (62.1 miles) above the Earth's sea level, and is commonly used to define the boundary between the Earth's atmosphere and outer space.*Atmosphericentry*is the transition from the vacuum of space to the atmosphere of any planet or other celestial body. The term is not used for landing on bodies which have no atmosphere such as the Moon.*Atmospheric reentry* refers to the return to an atmosphere previously left for space. Often the word "atmospheric" is dropped and the term *reentry* (or *re-entry*) is taken to mean *atmospheric reentry* in context.

The successful exploration of space requires a system that will reliably transport payload such as personnel and instrumental etc. into space and return them back to earth without subjecting them an uncomfortable or hazardous environment. In other words, the spacecraft and its payloads have to be recovered safely into the earth. The process of controlled reentry of vehicles which are intended to reach the planetary surface intact. TheVehicles that typically undergo this process include ones returning from orbit (spacecraft) and ones on exo-orbital (suborbital) trajectories (ICBM reentry vehicles, some spacecraft.) Typically this process requires special methods to protect against aerodynamic heating. Various advanced technologies have been developed to enable atmospheric reentry and flight at extreme velocities. We have seen the re-entry capsules and winged space vehicles approach the earth followed by safe landing. However, this could be accomplished only after considerable research in high speed aerodynamics and after many parametric studies to select the optimum design concept.

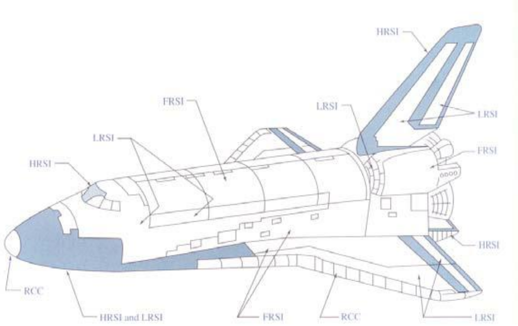
2. Re-Entry Mission Profile, Constraints and Vehicle Requirements

The safe recovery of the spacecraft and its payloads is made possible by the re-entry mission. According to the different constraints mission profile can be divided into three distinct flight segments:-

1) Deorbit and Descent to sensible atmosphere at an altitude of nearly 120kms.
2) Re-entry and hypersonic glide fight.
3) Transition flight phase, final approach and landing.
4) The various forces acting on the re-entry vehicle are:-
   - Gravitational force acting towards the centre of theplanet.
   - Gas dynamic force opposite to the direction of motion of the vehicle.
   - Centrifugal and gas dynamic lift force acting normal to the direction of
   - Motion of the vehicle

3. Thermal Protection Systems for Re-entry Vehicles

The vehicle's configuration and entry trajectory, in combination with the type of used surface material, define the temperature distribution on the surface vehicle.

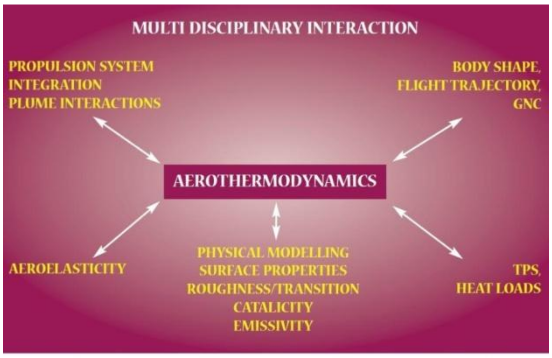


Space Shuttle Orbiter TPS

4. Literature Review

Aerothermodynamics is aimed at determining forces, moments and thermal loads of a spacecraft being launched or returning from orbit. The right shape as well as accurate prediction of the thermal loads determines the selection and dimensioning of thermal protection systems and load carrying structures, the guidance navigation and control of the vehicle as well as numerous mechanical and electrical subsystems. The aerothermodynamic behavior of space transportation systems needs to be verified by computations using advanced numerical tools, experimental investigations in ground based experimental facilities and by flight tests and is essential for the optimization of aerodynamic shapes

Aerothermodynamics is a key technology for the design and optimization of space vehicles because it provides the necessary databases for, for example, the choice of trajectory, for guidance, navigation and control, as well as for the thermal-protection and propulsion systems. Examples are presented of external flows past re-entry vehicle demonstrators and launchers. Internal flow problems associated with propulsion and the interactions with external flow arealso presented.



Interactions between aerothermodynamics and other disciplines

5. Same Basic Definitions Used In Case Of Aerothermodynmics

Heat flux: is the thermal power per unit area experienced by a TPS. The preferred units for heat flux are watts per square centimeter (W/cm²). The total heat flux experienced by an aero shell undergoing hypersonic entry can have up to three components, i.e. convective heat flux, catalytic heat flux and radiative heat flux.

Convective heat flux: is simply heat converted from the hot shock layer gas to the cooler aero shell wall.

Catalytic heat flux: is produced by dissociated gas species in the shock layer gas recombining into less reactive molecules on the aero shell wall thus releasing heat.

Radiative heat flux: comes from the intense light radiating from the shock layer which is in a state of chemical non-equilibrium due to passing through the shock wave. As a function of time from entry, radiative heat flux always reaches its peak value before the convective heat flux reaches its peak value (this can be used as a simple test of a heating model's validity).

The **Fay-Riddell equation** is a relatively compact closed form equation used to model the convective and catalytic heat flux at the stagnation point of an aero shell.[1] The Fay-Riddell equation is remarkably accurate and sometimes used to validate modern computational fluid dynamics (CFD) solutions.

Heat load: is time integrated heat flux.

Heat soak: is the component of heat load that actually penetrates the TPS and entry vehicle structure.

Heat pulse: The interval along the trajectory where the heat flux rises from insignificance, reaches its peak value and then descends back into insignificant is called the heat pulse. Heat pulse plotted as a function of time typically has a bell curve shape

6. Requirement of CFD in reentry ATD phenomena

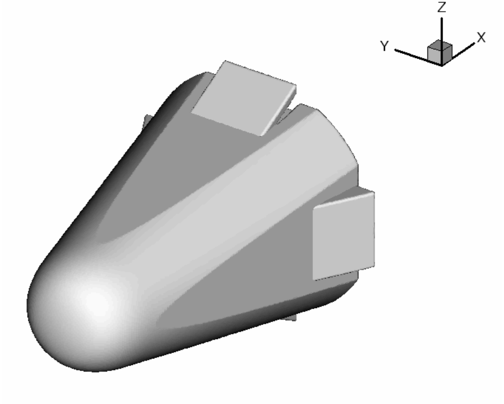
Flight experiments are typically associated with high costs, while numerical simulations containseveral shortcomings when modeling aero-thermo-dynamic processes. Therefore, long-duration,high-enthalpy, ground-test facilities are still the key tools for the design, verification andqualification of re-entry vehicles hot structures. During the on-ground experimentation, to achieve flight-like conditions we use arc-jet high enthalpy flows. The use of ground-based, high-enthalpy facilities such as SPES (Small Planetary Entry Simulator) requires a detailed knowledge of the test section flow. Because the flow conditions generated in high-enthalpy tunnels are very complex, the characterization of the free-stream flow must be a combined effort of measurement techniques and computational fluid dynamics (CFD) simulations

Hypersonic flight data are required for improved understanding of the following critical Aerothermodynamic (ATD) phenomena

1) Transition,
2) Catalicity and oxidation,
3) Real gas effects on shock wave boundary layer interactions,
4) Micro aerothermodynamics,

7. Introduction to the Expert Reentry Project

The configuration chosen for the EXPERT vehicle is basically a blunt cone with four flaps and flat surfaces ahead of them shown. EXPERT is expected to provide hypersonic flight conditions that could not be duplicated in existing hypersonic wind tunnels; in this way, a number of experiments will be carried out to study and validate aerothermodynamics models. The vehicle will be launched by the Russian Volna launcher and will make controlled ballistic suborbital flight to study the most critical aerothermodynamic phenomena encountered during atmospheric reentry, and then will be recovered for post flight inspection. A number of experiments will be carried out to study and validate aerothermodynamics models.

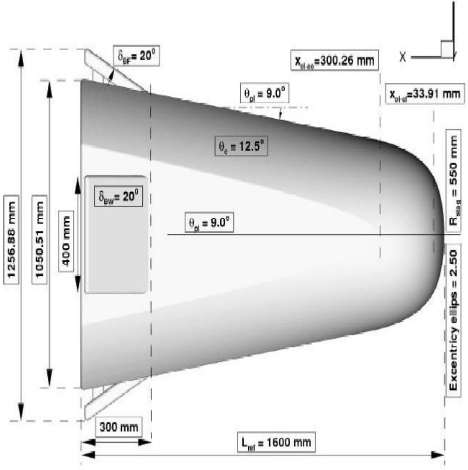


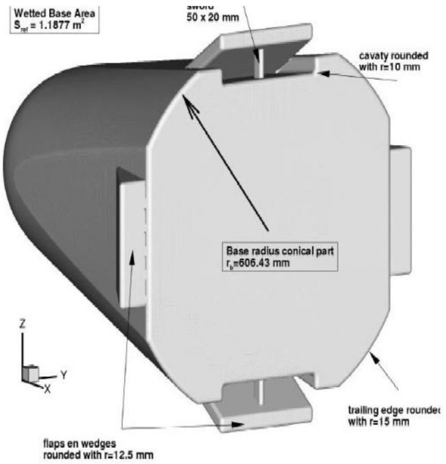
Expert Capsules

8. Modeling and Meshing of Expert Capsules

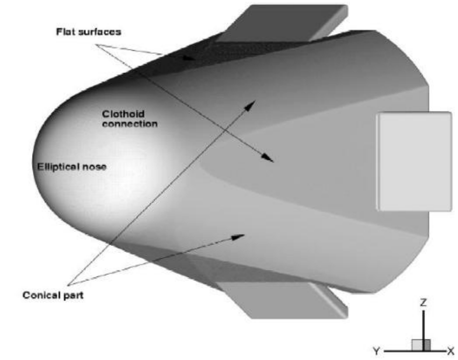
The solution procedure of this project work involves the following steps:
1 Modeling by using CATIA V5
2 Meshing by using GAMBIT
3 Flow analysis using FLUENT 6.3.16

Dimensions of an EXPERST Capusule





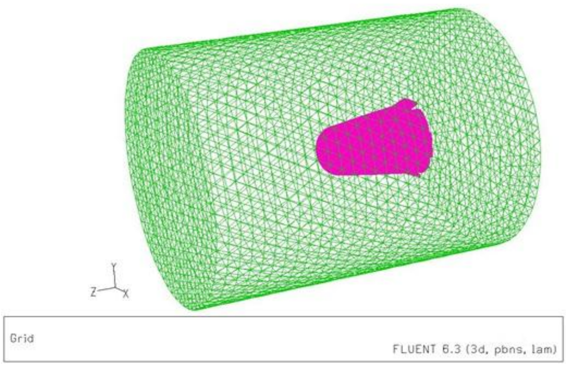
Side view
Figure
Base view



9. Meshing by Using Gambit

The solid model was imported from CATIA to GAMBIT. The model of the EXPERT Capusule is modified slightly to do the flow analysis. Here we are going to analysis the flow over the Capusule so we need the flow domain for the flow analysis. Therefore for flow analysis, a flow domain is created as for the dimensions required. Here we are creating the cylindrical shape domain with diameter 4L and length 7L.where L is the length of the Capusule Before starting of the mesh we need to create the boundary layer around the Capusule body. And then mesh the faces of the body by using tri pave mesh. To create 3D mesh of the domain we are using the tetrahedral pave elements. Then we are checking the mesh of the domain.

Perspective view
Meshed capsule within the flow domain



10. Flow Analysis Using Fluent 6.3.16

After creating and meshing of the capsule in Gambit then we are Exporting the model in to the FLUENT 6.3.16 for the flow analysis for the flowing references.

Mach number **Ma=15,**
The altitude is **45.7 *Km*,**
The static pressure is **P∞ = 130.5Pa,**
The stagnation pressure is **Pstang=39,000Pa**,
The static temperature is **T∞ = 267K,**
Theflow has an angle of incidence of three degree**α = 3**

After importing of the mesh file in to the FLUENT .we are checking the mesh for the accurate solution. After checking the mesh we are Define the initial condition such as Modeltype, material properties, Operating condition, and Boundary condition to the problem. Here we are taken Model Type Solver Density based. Because here the flow is compressible so the Density of the flow is very with time.

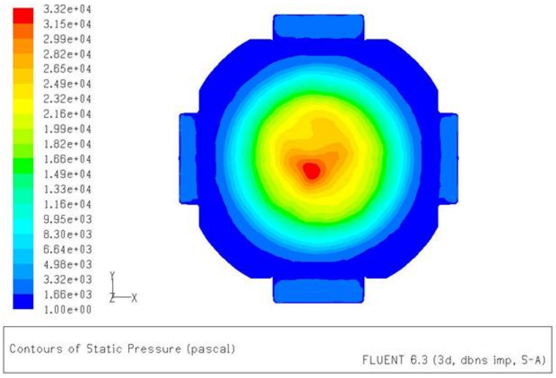
11. Results and Discussions

The physics of all this process is quite complex and it is clearly beyond the aim and the possibilities of the present project to approach the problem. For the purposes of present project, we chosen to adopt a model that can describe follow the phenomenology of problem avoiding to follow all the fine details. So that we adopted the Monti-Napolitano (MN) model

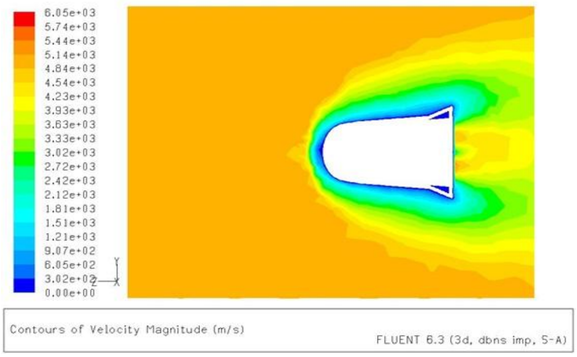
The design of EXPERTS capsule has a quite complex mission development. The capsule is released, starting the real re-entry trajectory. At a certain point of the trajectory the capsule is inflated realizing an increase of the nose radius. One of the key point of EXPERTS projects is the requirement of inherently security. In fact, if inflation is successful the capsule is decelerated and it will land without any risk, on the contrary, if for any reason the inflation fails, thermal protections should be designed in such a way that to heat flux produced will burn it in the atmosphere. Therefore it is very important to verify that thermal protection system can resist to the heat flux produced during re-entry, ensuring the complete destruction of the capsule.

During this first phase of the project an easy to use engineering formula is needed to make a first estimation of the heat flux, a more accurate prediction will be made in the following in correspondence of the critical situations by means of CFD simulations.

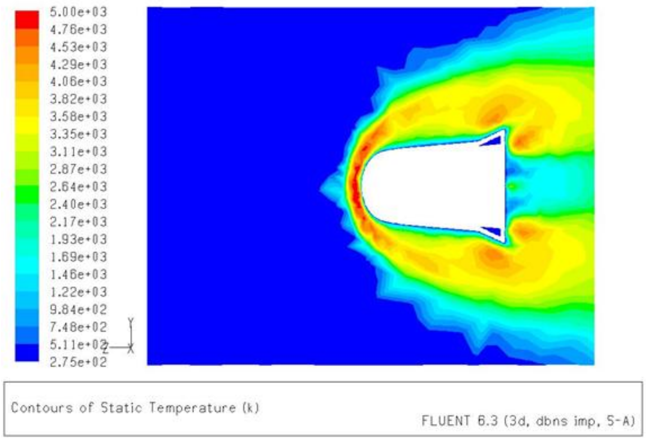
At Mach 15 the Countour Plots



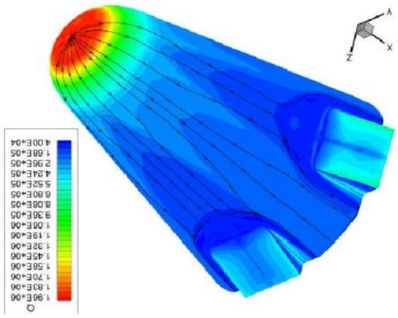
Static pressure distributionanEXPERTS capsule



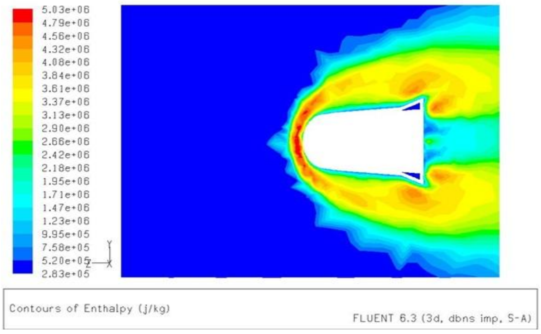
Velocity magnitude distributionover a capsule



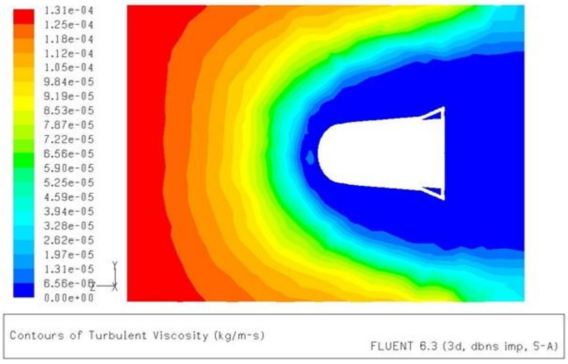
Static temperature distribution over a capsule



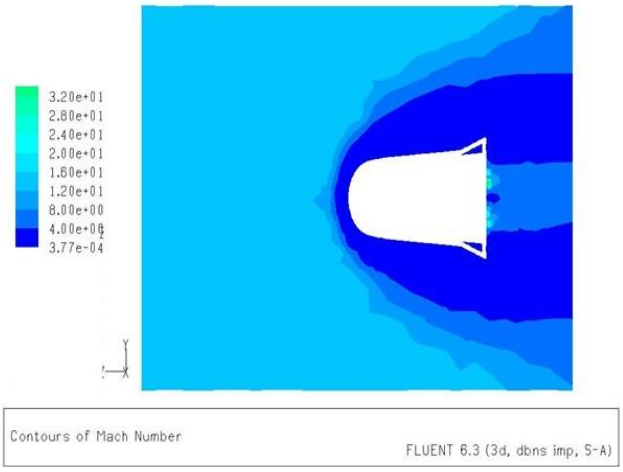
Heat flux distribution on a capsule



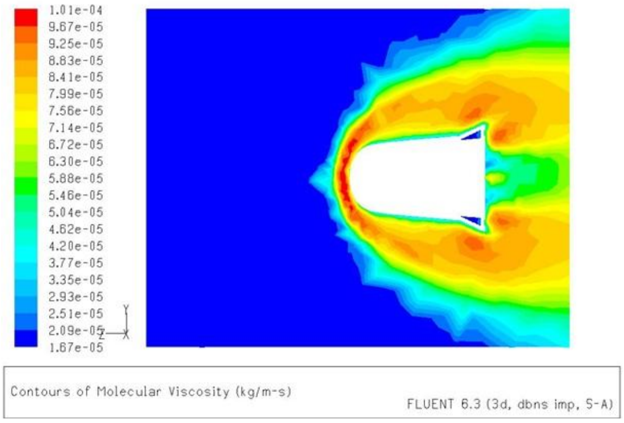
Enthalpy distribution over a capsule



Turbulent viscosity distribution over a capsule

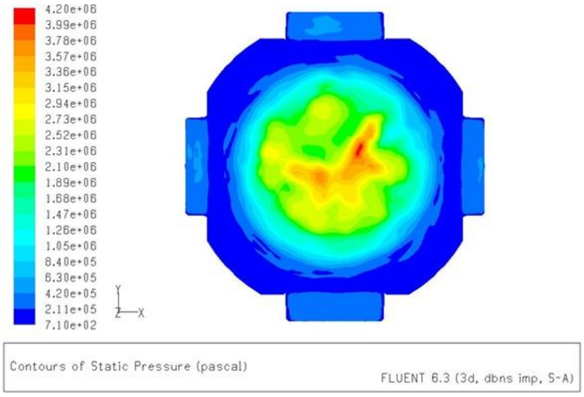


Mach number distribution over the capsule

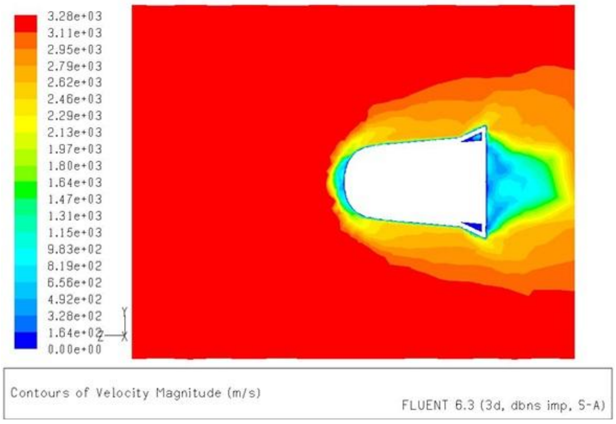


Molecular viscosity distribution over the capsule

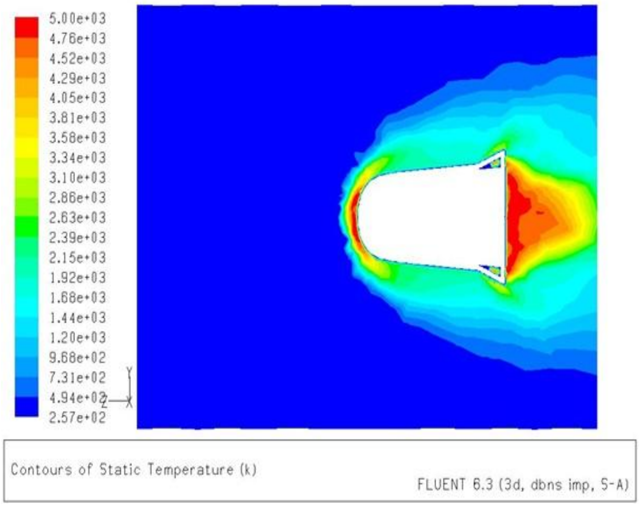
Figure at Mach 10 the Countour Plots



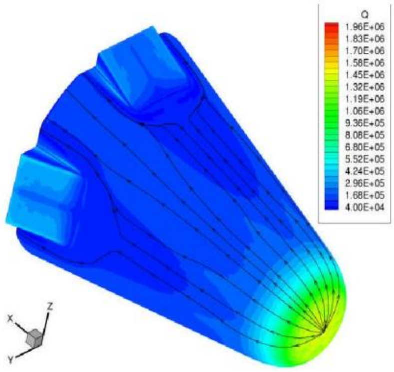
Static pressure distribution an the capsule body



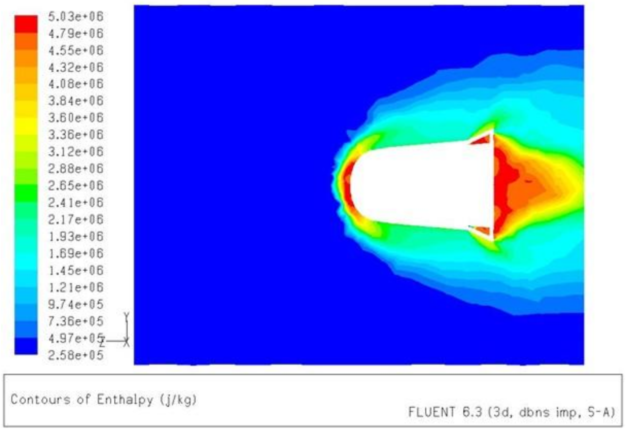
Velocity magnitude distribution over capsules



Static temperature distribution over the capsule



Heat flux distribution an the capsule



Enthalpy distribution over a capsule

In this flight condition, thermal protection system should be projected to resist to the heat fluxes developed during the re-entry path

Geometry	Mach number	q&[Present] (W/m2)	q&[anderson's5] (W/m2)
EXPERTS bluentshape capsule	15	1.96E06	2.03E06
EXPERTS bluent shape capsule	10	1.58E06	1.87E06

Comparison with the max heat fluxes on the body with Anderson's formula

Since the re-entry vehicle under study is completely new designed there is no possibility for any comparison, so for this reason we have presented the comparison with results obtained byAnderson's formula in order to make a check on the order of magnitude of heat flux. On the other hand it is clear that a maximum difference of 13% between an engineering formula and a CFD simulation is not surprising.

12. Conclusions

Numerical evaluation of convective heat flux on EXPERTS re-entry vehicle has been presented. With two different case of the EXPERTS vehicle have been considered and numerically simulated. A dissociative model, based on the Monti-Napolitano method, has been adopted to take into account the deceleration of EXPERTS capsule in high-energy conditions. The obtained results show a numerical difference that may appear not "small" when compared with other applications conducted in different, more common, flow conditions.For this reason, a possible alternative seems to inflate the capsule at higher altitude.

EXPERT is an in-flight research programme, with the objective to improve our understanding of critical aerothermodynamics phenomena such as transition, real gas effects and shock wave boundary layer interactions associated with flap efficiency and heating. At present one ballistic flight (5 km/sec) is planned. If successful, it is believed that other flights are possible such as for higher speed (6 km/sec); future flights for the study of transitional flow phenomena and skipping trajectories, jet interaction, test beds for MHD/nose heat flux reduction schemes and flights for the study of advanced materials associated with high-speed re-entry.

References

[1] EXPERT Aerodynamic and Aerothermodynamic analysis of the REV and KHEOPSconfigurations,
[2] Selected aerothermodynamic design problems of hypersonic flight vehicles By E.H. Hirschel , C. Weiland
[3] Anderson J.D., Hypersonic and high temperature gas dynamics, McGraw Hill, 1989.
[4] Muylaert, J-M.et al, Aerothermodynamic Environment of EXPERT and Flight.