Aerodynamic and aerothermodynamic analysis of high-speed earth re-entry capsule

Giuseppe Pezzella* and Eduardo Trifoni

Centro Italiano Ricerche Aerospaziali (CIRA),
via Maiorise, 81043 Capua, Italy
Email: g.pezzella@cira.it
Email: e.trifoni@cira.it
*Corresponding author

Abstract: This paper deals with the computational fluid dynamics analysis of a capsule entering the earth atmosphere at the end of a sample return mission. Several numerical simulations have been carried out to assess the flowfield environment past a sphere-cone capsule, by using non-equilibrium reacting gas approximations. Heat shield ablation and effects of flow plasma radiation on capsule aeroheating are also accounted for in the flowfield analysis. The issue of aerodynamic and aerothermodynamic performance of the entry spacecraft in the framework of a super-orbital re-entry scenario is addressed.

Keywords: computational fluid dynamics; flow field plasma radiation; heat shield ablation; hypersonic aerodynamics; hypersonic aerothermo-dynamics; hypersonic flow; reacting flow; sample returning mission; super-orbital re-entry.


Biographical notes: Giuseppe Pezzella received his Diploma (MSc) in Aeronautical Engineering in 1999 from the University of Naples ‘Federico II’. He took his PhD in Aerospace Engineering at University of Naples ‘Federico II’, in 2003. He has been research engineer in the Physics of Fluids Department at CIRA since 2005. He is a specialised analyst in the fields of vehicles, aerodynamics, aerothermodynamics, and aeroshape design. He has been an author or a co-author for about 100 publications among archive journals, conference papers, and edited books. He is reviewer and editorial/advisory board member of several international journals.

Eduardo Trifoni is the Test Engineering Group Leader of Plasma Wind Tunnel at CIRA. He is in charge of SCIROCCO 70 MW Plasma Wind Tunnel test campaign management, design, preparation, execution, reporting and evaluation. He is a Project Manager/Engineer of several past and on-going PWT test activities and Point of Contact for worldwide PWT Customers. He is involved in concept, design, procurement, follow-up, assembly, integration and testing of Thermal Protection Systems, Hot Structures, Actively Cooled Structures and Facility upgrades. He has worked in technology frontier fields broadening from hydrogen energy systems to hypersonics, performing fluid dynamic, thermal and mechanical skills in design, development, testing and evaluation.
Aerodynamic and aerothermodynamic analysis

1 Introduction

The capability to send vehicles into space with the aim of collecting samples from planets (i.e. Mars) and other celestial bodies represents an important step forwards for space exploration activities and for a more accurate knowledge of the earth and universe (Rivell, 2006). In this context, the design of a sample return vehicle (SRV) demands to solve strong technological design issues as the good understanding of the loading environment encountered by the spacecraft during the earth re-entry at super-orbital speed.

In order to make affordable sample return missions in the coming decade, Europe has planned to design several high-speed re-entry missions, crucial for planetary exploration (see Figure 1).

Figure 1  Future European atmospheric entry missions with high-speed entry (Adler, 2011) (see online version for colours)

Generally speaking, during a planetary re-entry, when a capsule or a spacecraft approaches the earth atmosphere, a strong bow shock detaches ahead of the vehicle. It is subjected to wide range of pressure, heat transfer and shear stresses. Several features specific of hypersonic regime appear as thin shock layer, entropy layer, viscous interaction, etc. (Howe, 1989; Mitcheltree et al., 1998; Viviani and Pezzella; Viviani and Pezzella, 2010). Among those phenomena, one could remark the high temperature effects, also known as real gas effects, since they appear as one of the critical points in the design phase of the vehicle mission (Viviani and Pezzella; Viviani and Pezzella, 2010).

Indeed, across the shock a large amount of kinetic energy is converted into thermal energy. This energy leads to high temperature of the gas mixture in the shock layer where dissociation and ionisation take place. It results into a plasma flow which impinges on the vehicle wall. To sustain this important heat-transfer, the spacecraft must be equipped with suitable thermal protection system (TPS), as those made of ablative material needed to accommodate very high-energy re-entry. Ablative heat shield is essential for the success of the re-entry manoeuvre, but their design is complex due to the nature of the heat-transfer involving aerothermodynamic phenomena as mass blowing of the heat shield species into the boundary layer and their interaction with dissociated air (Bertin, 1994; Scott, 1985; Viviani, Pezzella and Golia, 2010). In this framework, the capability to address computational flowfield analysis past a capsule returning at super-orbital speed is fundamental (Anderson, 1989; Viviani and Pezzella, 2012). In fact, these extra-terrestrial samples have to be returned to earth, and very high-speed re-entry trajectories must be performed by the space vehicles, according to celestial mechanics laws. Therefore, the design of an earth entry SRV requires strong technological bases and has to rely on a good understanding of the loading environment encountered during the super-orbital earth re-entry.
A high-speed earth entry vehicle has the following characteristics: entry velocity higher or equal to 11.7 km/s (compared to 7.5 km/s for the US Space Shuttle); very high heat fluxes (more than 10 MW/m²) and heat loads (in the range of 200 MJ/m²), where the radiative part is rather important. Indeed, the extreme environment in which the SRV must fly is recognised in Figure 2, where the range of velocities of interest together with thermo-chemical non-equilibrium flight regimes in earth’s atmosphere are also indicated (Howe, 1989).

**Figure 2** Comparison of thermo-chemical non-equilibrium flight regimes in earth’s atmosphere (Howe, 1989)

In this framework, the paper provides an overview of the research activities carried out by CIRA in the frame of the RAdiation-Shapes-Thermal protection investigAtionS for high-SPeed EAarth Re-entry (RASTAS SPEAR) study, founded by the 7th research framework (FP7/2007-2013). The main objective of the RASTAS SPEAR project was to increase Europe’s knowledge in high-speed re-entry vehicle technology to allow for planetary exploration missions in the coming decades.

In particular, CIRA was involved in CFD simulations carried out with the aim to assess the effects of: aeroshape modification, flowfield radiation, and of surface mass blowing on the SRV aerodynamics and aerothermodynamics. To this end, a SRV capsule configuration has been selected and a re-entry trajectory has been computed to address research analysis.

In the first design task, namely assessment of aeroshape modification, 12 CFD simulations were performed. They refer to two-dimensional (2D) Navier-Stokes (NS) computations in thermo-chemical non-equilibrium at given points along the re-entry trajectory, namely points M. This task foresees six simulations on non-ablated aeroshape and six on ablated aeroshape, thus assessing the effect of aeroshape modification due to the heat shield ablation.

In the second design task 12 2D Navier-Stokes computations in thermo-chemical non-equilibrium at given points along the re-entry trajectory, namely points R, were
performed to evaluate plasma radiation effect through the Europeans Space Agency (ESA) code PARADE (Smith et al., 2006).

Finally, in the last design task eight CFD simulations were planned. They refer to 2D Navier-Stokes computations in thermo-chemical non-equilibrium performed on non-ablated shape considering blowing species due to heat shield ablation. So that, the objective of this task was to assess the effect of blowing species on the flowfield past the re-entry vehicle.

2 Spacecraft configuration and flight scenario

Trade-off design analysis among hypersonic drag (heating), subsonic drag (impact velocity) and subsonic stability (available crush stroke) suggests to consider a spherico-conical capsule aeroshape with a 45 deg half cone angle, a 1.1 m diameter front shield, a nose radius equal to 0.275 m, a shoulder radius of curvature equal to 0.0275 m and a smaller back-cover (see Figure 3) (Pezzella, Catalano and Bourgoing, 2012). The spacecraft, including margins, is estimated to weigh about 50 kg with the centre of gravity (CoG) at 26.9% from the nose relatively to the SRV diameter.

Figure 3 Capsule geometry with quotes (Pezzella, Catalano and Bourgoing, 2012) (see online version for colours)

The layout and design of the capsule (see Figure 4) assure a safe return of the sample canister, relying on a fully passive concept. The stability requirement of such a shape (CoG < 26.5%) is the main critical parameter in this SRV design.

Figure 4 SRV layout (see online version for colours)
The flight design scenario of the SRV concept is shown in Figure 5. It refers to an earth descent characterised by −12.5 deg of flight path angle (FPA), 12.3 km/s of entry velocity \( V_e \) at 120 km altitude and, therefore, by a rather high-heat flux peak. The evolutions of heat fluxes (convective and radiative part) with the corresponding integrated heat load are also presented in Figure 5.

**Figure 5** Re-entry trajectory in the altitude-velocity map and aerotherm al loading conditions (see online version for colours)

In the preliminary design, the convective and radiative heat fluxes were estimated by using analytical engineering correlations such as Scott relationship for convective heat flux and Tauber-Sutton one for the radiative heat flux evaluation (Scott et al., 1985; Tauber and Sutton). All the preliminary estimations provided in Figure 5 are extremely important for designing the capsule. Indeed, the descent trajectory provides initial conditions for CFD solutions while the aeroheating environment dictates the type and size of the TPS to be used. Peak heat rate generally determines the range of possible thermal protection material (TPM) while the integrated heat load determines the thickness and hence the mass of the capsule heat shield.

The time histories of heat fluxes (convective and radiative parts), Mach number, altitude and velocity are presented in Figure 6.

**Figure 6** Heat Flux, Mach, altitude, velocity versus time (see online version for colours)
The evolutions of the deceleration, dynamic pressure, altitude, velocity and FPA versus time are presented in Figure 7. As shown, at the peak dynamic pressure is equal to about 44578 Pa; while the maximal aerodynamic deceleration is less than 70 g.

**Figure 7** Deceleration, dynamic pressure, altitude, velocity and FPA versus time (see online version for colours)

The materials chosen for the heat shield (see Figure 4) is the ASTERM (a low density carbon phenolic material) for the front-shield, and the Norcoat-Liège (a cork based material) for the back-shield. The ASTERM and the Norcoat-Liège were shown well adapted to the aerothermal and mechanical stress of the mission. The maximum TPS thickness need is sized with 1D calculation on the more thermally severe conditions with regard to a maximum allowable criterion of 150°C for the bonding material. To simplify the design and to tend towards a more robust configuration, the thickness on both the front-shield and the back-shield were considered constant. This leads to the following thickness: front-shield (FS TPS): 56 mm and back-shield (AFT TPS): 8 mm.

### 3 Aerodynamic and aerothermodynamic performance analysis

The appraisal of the aerodynamic and aerothermodynamic characteristics of the SRV concept is performed.

These evaluations are aimed to address the aerothermal loads the vehicle has to withstand during re-entry at super-orbital speed. To this aim, a number of extreme loading flight conditions have been focused along with the sizing flight trajectory, according to the trajectory-based design approach (Viviani and Pezzella). The heat flux distributions, both convective and radiative one, are also addressed for each chosen trajectory design point. However, it is worth noting that due to the prohibitive computational effort, no fully coupled computations of heat shield ablation and flow plasma radiation were undertaken in this work.

In the present analysis, only continuum regime (between Mach 3 and Mach 41.54) with the air modelled as a mixture of several gases (including also those coming from heat shield ablation) has been analysed.
In particular, the appraisal of the vehicle aerodynamic and aero-thermodynamic was performed by means of both engineering-based tools and Navier-Stokes CFD computations in thermo-chemical non-equilibrium and both in laminar and turbulent flow conditions. Then, results of CFD simulations are provided to PARADE code for the estimation of the radiative heat fluxes the capsule has to withstand during the super-orbital re-entry flight.

Engineering-based aerodynamic analyses were extensively performed by using a 3D panel method code developed by CIRA, namely SIM (Surface Impact Method) (Viviani and Pezzella; Viviani and Pezzella, 2012). This tool at high supersonic and hypersonic speeds is able to accomplish the aerodynamic and aero-thermodynamic analyses of a complex re-entry vehicle configuration by using simplified approaches as local surface inclination methods and approximate boundary-layer methods, thus avoiding the time consuming and complex grid generation phase and the computation processes of a CFD analysis.

The numerical tool used to carry out the CFD analyses is the CIRA code H3NS/CAST (Viviani and Pezzella, 2010, 2012). It solves the flowfield governing equations, including chemical and vibrational non-equilibrium, with a finite volume approach. The fluid is treated as a mixture of perfect gases and the energy exchange between vibrational and translational modes (TV) is modelled with the classical Landau-Teller non-equilibrium equation, with average relaxation times taken from the Milikan-White theory modified by Park. As far as the transport coefficients is concerned, the viscosity of the single species is evaluated by a fit of collision integrals calculated by Yun and Mason, the thermal conductivity is calculated by means of the Eucken law; the viscosity and thermal conductivity of the gas mixture are then calculated by using the semi-empirical Wilke formulas. The diffusion of the multi-component gas is computed through a sum rule of the binary diffusivities of each couple of species (from the tabulated collision integrals of Yun and Mason).

Transport coefficients, in the hypothesis of an ideal gas, are derived from Sutherland law, suitably modified to take into account for low temperature conditions. With respect to the numerical formulation, conservation equations, in integral form, are discretised with a finite volume, cell centred technique. Convective fluxes are computed with a flux difference splitting (FDS) upwind scheme. Second-order formulation is obtained by means of an ENO-like reconstruction of cell interface values. Viscous flux is computed with a classical centred scheme i.e. computing the gradients of flow variables at cell interfaces by means of Gauss theorem. Integration in time is performed by employing an explicit multistage Runge-Kutta algorithm coupled with an implicit evaluation of the species and vibration energies source terms. Also a parallel version of the code is currently available.

The mathematical formulation describing a flowfield around a hypervelocity vehicle deals with balance equations for a multi-species chemically reacting gas mixture.

### 3.1 Flowfield governing equations

The mathematical model is made up of the equations for mass conservation, momentum balance, total energy (less vibrational one) conservation, individual species balance and vibrational energy conservation. The full set of equations for a viscous compressible continuum flow in thermal and chemical non-equilibrium, assuming the air to be a
Aerodynamic and aerothermodynamic analysis

A mixture of $N_a$ perfect gases (mixture species) and $N_v$ vibrating species, can be written in the integral conservation form as follows:

$$
\frac{\partial}{\partial t} \int_V \dot{\rho} \, dV + \int_V \left( \frac{\partial}{\partial t} \left( \dot{\rho} \dot{V} \right) \right) + \nabla \cdot \left( \dot{\rho} \dot{V} \right) + \dot{\rho} \dot{V} = \int_V \dot{\Omega} \, dV
$$

(1)

where $\dot{\rho} = \left[ \rho, \rho u, \rho v, \rho w, e_1, \rho_1, \ldots, \rho_{N-v}, \rho_{e_{v1}}, \ldots, \rho_{e_{vN_v}} \right]^T$ is the unknown state vector of the conserved quantities, in which $\rho$ is the fluid density; $\rho u$, $\rho v$, and $\rho w$ are the momentum densities; $e_1$ is the total internal energy per unit mass; $\rho_1$ and $e_{v1}$ are, respectively, the density and the vibrational energy of the $i^{th}$ species, while $\rho_{e_{v1}}$ takes into account for vibrational energy conservation; and $\dot{F}$ is the flux vector (split into an inviscid and a viscous part). The vector $\dot{F}$ expression is well known in literature and therefore it is not written here for simplicity. $\dot{\Omega}$ is the axis-symmetric terms matrix (split into an inviscid and a viscous part), and $\Gamma$ is equal to 1 for axis-symmetric flows and 0 for 2D and 3D flows. $\dot{\Omega} = \begin{bmatrix} 0, 0, 0, 0, 0, \ldots, \Omega_{N-v1}, \Omega_{e_{v1}}, \ldots, \Omega_{e_{vN_v}} \end{bmatrix}^T$ is the source terms vector that defines the mass and energy exchange among the species as a result of the chemical reaction rate and the energy transfer due to the internal energy excitation processes. Hence, $\dot{\Omega}$ models the non-equilibrium reactions.

Finally, $V$ is the arbitrary control volume cell, $S$ is its closed boundary control surface and $\hat{n}$ is the outward normal unit vector. Equation (1) can be written in differential form as follows:

Continuity:

$$
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \dot{V}) = 0
$$

(2)

Momentum:

$$
\frac{\partial (\rho \dot{V})}{\partial t} + \nabla \cdot (\rho \dot{V} \dot{V}) + \dot{\nabla} \rho = 2 \nabla \left[ \mu (\nabla \dot{V})^T \right]
$$

(3)

where

$$
(\nabla \dot{V})^T = \frac{1}{2} \left[ (\nabla \dot{V}) + (\nabla \dot{V})^T \right] - \frac{1}{3} (\nabla \cdot \dot{V}) \nabla
$$

(4)

Energy:

$$
\frac{\partial (\rho e_i)}{\partial t} + \nabla \cdot \left( \rho e_i \dot{V} \right) = \nabla \left[ \lambda (\nabla \cdot \dot{V}) + 2\mu (\nabla \dot{V})^T \cdot \dot{V} + \sum_j h_j \dot{J}_j \right] - \sum_j h_j \dot{\omega}_j - \sum_j \dot{\epsilon}_{v_j}
$$

(5)

where

$$
\dot{\epsilon}_{v_j} = \left( e_{v_j} - e_{v_j} \right) / \tau_i
$$

(6)

Species:

$$
\frac{\partial (\rho Y_j)}{\partial t} + \nabla \cdot \left( \rho Y_j \dot{V} \right) + \dot{\nabla} \cdot \dot{J}_j = \dot{\omega}_j
$$

(7)
where
\[ \dot{\omega}_i = M_i \sum_k \dot{\omega}_k \] (8)

Vibrational energy:
\[ \frac{\partial (\rho e_{i_v})}{\partial t} + \nabla \cdot (\rho \vec{v} e_{i_v}) = \dot{\varepsilon}_i \] (9)

In these equations, \( \vec{V} \) is the velocity vector, \( Y_i \) is the mass fraction of the \( i \)th species and \( \dot{\omega}_i \) is the rate of change of \( \rho_i \) due to chemical reactions, \( J_i \) is the diffusive flux of \( i \)th species, which arises due to concentration gradients, \( M_i \) and \( h_i \) are, respectively, the molecular weight and enthalpy of \( i \)th species, \( p \) is the pressure, \( U \) is the unit tensor, \( \mu \) is the viscosity and \( \lambda \) is the thermal conductivity.

For each species, the perfect gas model applies and the Dalton’s law is used:
\[ p = \sum_i p_i \] (10)

where \( p_i \) is the partial pressure of the \( i \)th species of the mixture.

As a consequence, the following relation for density holds:
\[ \rho = \frac{p}{R_0 T \sum Y_i / M_i} \] (11)

where \( R_0 = 8314.5 \text{ J Kmol}^{-1} \text{ K}^{-1} \) is the universal gas constant.

The internal energy of the mixture is defined as:
\[ e = \sum_i (Y_i e_i) \] (12)

where \( e_i \), internal energy of the single component gas, is the sum of the energies representing the different degrees of freedom of the molecules. Finally, the specific enthalpy for each species can be calculated as:
\[ h_i = e_i + R_i T \] (13)

where \( R_i \) is the gas constant.

The flow radiative heat flux at the SRV wall has been computed through the code PARADE, starting from the results of the fluid dynamic computations (in terms of gas composition and temperature).

This code is able to compute flow-field emission and absorption, between the shock layer and the surface of the probe. The spectral emission and absorption are determined as function of transition level (from upper level to lower level) and emitting population of this level. The population can be derived from the quasi-steady-state (QSS) method or by a Boltzmann method in order to take into account for the non-equilibrium or equilibrium regime, respectively. The radiative computations have been performed with the Boltzmann assumption for the determination of the population of the excited molecular states. The radiative heat transfer equation (RTE) has been solved using the one-dimensional tangent slab approximation (radiation properties are assumed to vary only in
the direction normal to the wall). The intensity of radiation at a given wavelength $\lambda$ satisfied the equation of radiative transfer:

$$\frac{\partial I_s}{\partial s} = j_\lambda - k_\lambda I_s$$  \hspace{1cm} (14)

where $j_\lambda$ and $k_\lambda$ are, respectively, the emission and absorption coefficients, computed through PARADE. These coefficients are integrated along with straight lines towards the wall according to the above equation to compute $I_s$.

Radiative heat flux at a given wavelength $\lambda$ is then obtained through integration of $I_s$ over the solid angle whereas the total radiative heat flux is obtained through integration over the spectrum of interest. In particular, a spectral region between 100 and 40000 nm using 50000 spectral grid locations has been considered in the calculations.

4  Loading scenario and air mixture composition

SRV aerothermal analyses refer to CFD simulations performed at the several discrete points of the flight scenario summarised in Figure 8 (Pezzella, Catalano and Bourgoing, 2012). Furthermore, to get an idea of real gas effects the capsule will experience during descent, Figure 8 also shows the re-entry trajectory superimposed on the fields (from 10% to 90 %) of vibrational excitation (VE), oxygen and nitrogen dissociation (OD and ND), and ionisation (I) of flow species (Viviani and Pezzella; Viviani, Pezzella and Golia, 2010).

Figure 8  Re-entry trajectory in the altitude-velocity map with design point (Pezzella, Catalano and Bourgoing, 2012) (see online version for colours)

The assessment of aeroshape modification on both aerodynamic and aerothermodynamic is determined by means of 12 2D axisymmetric NS simulations in thermo-chemical non-equilibrium carried out at six given points along the re-entry path. These trajectory points have been selected by considering the heatshield recession level occurring at the vehicle stagnation point during descent, as summarised in Figure 9.
As one sees, the six points M have been chosen according to the increase of recession level. Moreover, the recession increases rapidly during the first 50s of the trajectory but still remain a half (about 5.5 mm) compared with the final recession level of 10 mm. Points M are plotted in the altitude-velocity map, as presented in the Figure 10; while the free stream conditions are summarised in the Table 1.
### Table 1
Free stream conditions of points M (yellow: laminar regime, pink: turbulent regime) (see online version for colours)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>M1&amp;M1a</td>
<td>24.8</td>
<td>58.73</td>
<td>11099</td>
<td>26.49</td>
<td>3.5732 × 10$^{-4}$</td>
<td>258.26</td>
<td>34.45</td>
<td>1.3</td>
</tr>
<tr>
<td>M2&amp;M2a</td>
<td>28.2</td>
<td>52.05</td>
<td>9604</td>
<td>61.84</td>
<td>7.9596 × 10$^{-4}$</td>
<td>270.65</td>
<td>29.12</td>
<td>2.6</td>
</tr>
<tr>
<td>M3&amp;M3a</td>
<td>30.4</td>
<td>48.36</td>
<td>8280</td>
<td>97.84</td>
<td>1.2594 × 10$^{-3}$</td>
<td>270.65</td>
<td>25.11</td>
<td>3.2</td>
</tr>
<tr>
<td>M4&amp;M4a</td>
<td>34.4</td>
<td>43.14</td>
<td>5681</td>
<td>189.53</td>
<td>2.5491 × 10$^{-3}$</td>
<td>259.02</td>
<td>17.61</td>
<td>4.0</td>
</tr>
<tr>
<td>M5&amp;M5a</td>
<td>43.2</td>
<td>36.93</td>
<td>2181</td>
<td>437.43</td>
<td>6.3007 × 10$^{-3}$</td>
<td>241.86</td>
<td>7.00</td>
<td>5.1</td>
</tr>
<tr>
<td>M6&amp;M6a</td>
<td>53.2</td>
<td>33.62</td>
<td>922</td>
<td>701.11</td>
<td>1.0497 × 10$^{-23}$</td>
<td>232.69</td>
<td>3.02</td>
<td>5.3</td>
</tr>
</tbody>
</table>

The eight trajectory points (i.e., points B) for the assessment of heat shield ablation effects are selected by the examination of mass blow rate versus time, reported in Figure 11. As shown, the major part of these points is around the maximum value of total mass blow rate.

**Figure 11** Mass blow rate versus time (ablation and pyrolysis contribution) (see online version for colours)

![Mass blow rate versus time](image)

Points B are plotted in the altitude-velocity map in Figure 12; while free stream conditions are summarised in Table 2.

Finally, design points (i.e., points R) considered assessing SRV aerodynamic and aerothermodynamic performance within the assessment of radiation coupling effect are depicted in Figure 13.
Figure 12  The eight selected points plotted in the Altitude/Velocity diagram-Blowing assessment (see online version for colours)

Table 2  Free stream conditions of points B (yellow: laminar regime, pink: turbulent regime) (see online version for colours)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>B1</td>
<td>19.0</td>
<td>71.86</td>
<td>12138</td>
<td>4.14</td>
<td>6.7859x10$^{-3}$</td>
<td>212.41</td>
<td>0.043</td>
<td></td>
</tr>
<tr>
<td>B2</td>
<td>22.4</td>
<td>63.98</td>
<td>11711</td>
<td>13.19</td>
<td>1.8886x10$^{-4}$</td>
<td>243.29</td>
<td>0.076</td>
<td></td>
</tr>
<tr>
<td>B3</td>
<td>25.6</td>
<td>57.07</td>
<td>10816</td>
<td>32.84</td>
<td>4.3738x10$^{-4}$</td>
<td>261.52</td>
<td>0.120</td>
<td></td>
</tr>
<tr>
<td>B4</td>
<td>28.2</td>
<td>52.05</td>
<td>9604</td>
<td>61.84</td>
<td>7.9596x10$^{-4}$</td>
<td>270.65</td>
<td>0.083</td>
<td></td>
</tr>
<tr>
<td>BE</td>
<td>30.4</td>
<td>48.36</td>
<td>8280</td>
<td>97.84</td>
<td>1.2594x10$^{-3}$</td>
<td>270.65</td>
<td>0.071</td>
<td></td>
</tr>
<tr>
<td>B6</td>
<td>34.4</td>
<td>43.14</td>
<td>5681</td>
<td>189.53</td>
<td>2.5491x10$^{-3}$</td>
<td>259.02</td>
<td>0.062</td>
<td></td>
</tr>
<tr>
<td>B7</td>
<td>39.6</td>
<td>38.85</td>
<td>3184</td>
<td>335.8</td>
<td>4.7331x10$^{-3}$</td>
<td>247.16</td>
<td>0.040</td>
<td></td>
</tr>
<tr>
<td>B8</td>
<td>43.2</td>
<td>36.93</td>
<td>2181</td>
<td>437.43</td>
<td>6.3007x10$^{-3}$</td>
<td>241.86</td>
<td>0.019</td>
<td></td>
</tr>
</tbody>
</table>

Figure 13  The six selected points plotted in the altitude/velocity map-plasma radiation effect (see online version for colours)
Points R have been selected following the evolution of the convective, radiative and, therefore, total heat flux as shown in Figure 14. Freestream conditions are summarised in Table 3.

**Figure 14** Heat flux evolution versus trajectory time-plasma radiation effect (see online version for colours)

**Table 3** Free stream conditions of points R

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>R1&amp;R1c</td>
<td>190</td>
<td>71.86</td>
<td>4.14</td>
<td>6.7859x10$^{-3}$</td>
<td>212.41</td>
<td>41.54</td>
<td>55</td>
<td></td>
</tr>
<tr>
<td>R2&amp;R2C</td>
<td>224</td>
<td>63.98</td>
<td>13.19</td>
<td>1.8886x10$^{-4}$</td>
<td>243.29</td>
<td>37.45</td>
<td>95</td>
<td></td>
</tr>
<tr>
<td>R3&amp;R3C</td>
<td>24.8</td>
<td>58.73</td>
<td>26.49</td>
<td>3.5732x10$^{-4}$</td>
<td>258.26</td>
<td>34.45</td>
<td>11.5</td>
<td></td>
</tr>
<tr>
<td>R4&amp;R4C</td>
<td>264</td>
<td>55.46</td>
<td>40.33</td>
<td>5.3082x10$^{-4}$</td>
<td>264.69</td>
<td>32.16</td>
<td>11.1</td>
<td></td>
</tr>
<tr>
<td>R5&amp;R5C</td>
<td>282</td>
<td>52.05</td>
<td>61.84</td>
<td>7.9596x10$^{-4}$</td>
<td>270.65</td>
<td>29.12</td>
<td>8.98</td>
<td></td>
</tr>
<tr>
<td>RB&amp;R6C</td>
<td>304</td>
<td>48.36</td>
<td>97.84</td>
<td>1.2594x10$^{-3}$</td>
<td>270.65</td>
<td>25.11</td>
<td>6.77</td>
<td></td>
</tr>
</tbody>
</table>

The atmosphere composition considered is made of: $Y_{N_2} = 0.75548; Y_{O_2} = 0.23161$ and $Y_{Ar} = 0.01291$. Therefore, the specific gas constant for air is $R_{air} = R_0/M = 287.182$ J/Kg/K. The model proposed for the air mixture is constituted by 13 species: Molecules: N$_2$, O$_2$, NO; Atoms: N, O, Ar; Ions: N$^+_1$, O$^+_1$, NO$^+$, N$^+$, O$^+$, Ar$^+$ and e$^-$; while to account for heat shield blowing mass the air mixture consists of 32 species: Molecules: N$_2$, O$_2$, NO, C$_2$, H$_2$, C$_3$, C$_2$H$_2$, C$_2$H, CO$_2$, H$_2$O, CN, CH, NH, HCN, OH, CO; Atoms: N, O, Ar, C, H; Ions: N$^+_2$, O$^+_2$, NO$^+$, N$^+$, O$^+$, CN$^+$, CO$^+$, C$^+$, H$^+$, Ar$^+$ and e$. The chemical model is based on Park (1993). It is constituted, when mass blowing is off, of 13 species and 22 chemical reactions that account also for third body efficiency. When heat shield species diffuse in the shock layer, the chemical model is constituted of 67 chemical reactions which feature also the effect of collisional partner. For what concerns the thermal non-equilibrium model it is worth to note that a three-temperature model is recommended: $(T, T_v, T_e)$ (Bertin, 1994; Anderson, 1989). $T$ for translational and
rotational mode for heavy species; $T_v$ for vibrational modes and $T_e$ for translational mode of free electron. In the present CFD computations, however, only two temperatures model ($T$ and $T_{vib}$ for each molecule of reacting mixture, i.e., $T_{v_N}$, $T_{v_O}$, $T_{v_NO}$, $T_{v_NO_2}$) has been considered.

Finally, flight regime assessment was made in order to be sure that all trajectory design points lie within the continuum flow regime to be able to use CFD analysis, as discussed in the next paragraph.

4.1 Flight regime overview and flow simulation modelling

Once SRV vehicle started its descent, the atmospheric density is low enough that the molecular mean free path ($\delta$) can become as large as the scale of the body itself. As a consequence, the continuum assumption does not apply. This condition is known as free molecule flow (FMF) regime and, the aerodynamic and aerothermodynamic characteristics of the capsule are determined by individual, scattered molecular impacts, and must be analysed on the basis of Kinetic Theory. Therefore, the SRV concept, moving from a very rarefied atmosphere (at entry interface) to a denser atmosphere, shifts from the FMF regime, where individual molecular collisions are important, to the transitional one, where slip effects are important, and then to the continuum regime, as represented in Figure 15.

Figure 15 SRV re-entry trajectory in the Ma-Re map with constant Knudsen numbers

For instance, the similarity parameter that governs these different flow regimes is the Knudsen number (rarefaction parameter):

$$Kn_{ref} = \frac{\delta}{L_{ref}} = 1.25\sqrt{\frac{\gamma}{Re_{ref}}} \frac{M_{\infty}}{Re_{ref}}$$

(15)

where $L_{ref}$ (i.e., 1.1 m-capsule diameter) is the characteristic length of the body. As shown in Figure 15, the maximum Mach number (i.e., $M_{\infty}=41.54$) evaluated by the SRV Flight Mechanics analysis lies in the continuum flow regime. Therefore, all CFD
computations have been performed with Navier-Stokes approximation without slip conditions at wall.

As far as laminar-to-turbulent flow transition is concerned, CFD computations are carried out in turbulent flow conditions for points below 51.9 km altitude, according the ARD post-flight data criterion (Viviani and Pezzella, 2012).

\[
\text{Re}_{\infty, D} = \left( \frac{\rho u D}{\mu} \right) \geq 5.0 \times 10^5
\]  

(16)

This criterion, however, represents only a first viable option considering that the effect of higher Mach number of the SRV trajectory compared to that of ARD and the mass blowing effect, as well, should influence departure from laminar boundary layer flow conditions at lower altitude. Furthermore, transition can be triggered by roughness due to ablation and/or gap and step of heat shield.

4.2 Computational domains, boundary conditions and solution convergence

CFD computations have been carried out on multiblock structured grids similar to those shown in Figure 16. As shown, 3D and 2D axis-symmetric computational grid for both whole and front shield domains have been considered. The effective dimensions of the outflow boundary, axis and outer boundary are modified in each simulation in order to obtain a grid compliant to the flow-field conditions. The strategy adopted was to build the volume mesh be large enough to accommodate the freestream Mach number; while the distribution of grid points in the wall-normal direction is driven by freestream Reynolds number. Hence, the distribution of surface grid points was dictated by the level of resolution desired in various areas of vehicle such as stagnation region and base fillet, according to the computational scopes. For example, the grid has sufficient points in the shoulder region to capture the rapid expansion and accurately predict the flow separation point and the angle of the resulting shear layer. There are also enough points in the separated flow region to resolve the vortical structure at the beginning of the wake flowfield. Three-dimensional computations are carried out on a mesh domain made of 48 blocks with about 800000 cells. The generic 2D whole domain grid has consisted of about 30 blocks for an overall number of about 72000 cells (half body). The grid (front shield case) is constituted by 124×160 cells (longitudinal × normal to the wall direction) and assures fully spatially converged results. A local refinement has been done in the shock region in front of the capsule to better resolve the steep gradients, aligning the grid with the bow shock and clustering points into the boundary layer. This reduced the spurious oscillations in the stagnation area that are often observed in hypersonic flows, especially for large bluff-body flowfield computations. The minimum spacing at the wall is equal to \(10^{-6}\) m to accurately predict heat transfer at the vehicle surface. When the flow is in turbulent conditions the values of the viscous coordinate \(y'\) are less than 1.

The SRV aeroshape to consider in mesh generations for numerical simulations regarding aeroshape modification effect are summarised in Figure 17.

As shown, there are slight aeroshape modifications compared with the initial profile for the selected trajectory points. The most important aeroshape change occurs in the nose and tore parts (the cone angle does not change). Therefore, only limited impact on aerodynamic coefficients and on heat fluxes are expected.
Figure 16  CFD multiblock computational domains (see online version for colours)

Figure 17  SRV aeroshape modification due to TPS ablation (see online version for colours)
As far as wall boundary conditions are concerned, all simulations without heat shield ablation (i.e., points M and R) are performed with a wall temperature of $T_w = 1000$ K (i.e., isothermal wall) and fully catalytic wall (FCW) assumption, i.e. chemical equilibrium at the wall, namely local equilibrium wall. For instance, as atoms produced by flow dissociation strike the surface, the catalytic property of the wall is implemented by means of a production term (i.e., $\dot{\omega}_w \neq 0$) for the boundary condition of the boundary layer problem to solve. Indeed, steady-state mass atomic conservation at the wall states that the production of $i$th species due to the catalytic recombination rate must be balanced by the rate of diffusion to the surface:

$$\left( \dot{\omega}_w \right)_i = k_{aw} \left( \rho_w Y_{aw} \right)_w = \left( \rho D_a \frac{\partial Y_a}{\partial n} \right)_w$$

where $\delta$ is the reaction order. When the TPS surface is FCW (i.e., activates all the recombination reactions at its surface) means that $k_{aw} = \infty$. As a result, a complete recombination of flow dissociated species takes place at wall because the flow locally tends towards chemical equilibrium conditions. In this case, the molecular species concentrations at the wall have to be set equal to their equilibrium concentrations according to the local temperature and pressure. For wall temperatures below 2000 K (i.e., cold walls) this corresponds to the freestream composition.

When heat shield mass diffuses in the shock layer (i.e., points B), numerical simulations are carried out considering a prescribed mass fraction profile at wall for each blowing species. The ablation model for ASTERM is based on thermo-chemical equilibrium oxidation of carbon as a function of surface temperature, pressure and pyrolysis gas mass flow which exercise a blocking effect on heating flux. Therefore, the heat shield recession to consider at each CFD computation was provided by ASTRIUM by means of suitable boundary conditions to consider at the capsule wall, as shown in Figure 18.

**Figure 18** Mass fraction profiles at $H = 63.98$ km altitude and temperature profile at different altitudes (Pezzella, Catalano and Bourgoing, 2012) (see online version for colours)

As one can see, wall temperature, mass blow rate and composition of injected products are variable along the capsule centreline and re-entry trajectories. Indeed, for the charring ablator such as a carbon-phenolic material, the reaction at the surface itself is divided into
oxidation dominant regime below 3000 K of surface temperature and sublimation dominant regime for higher temperature. In the latter regime, the surface recession rate increases rapidly with the temperature rise and a massive flow of ablation product must be taken into consideration in addition to the pyrolysis gassing.

All CFD results refer to both converged and grid independent computations. In order to assess numerical solution convergence, equation residuals and aerodynamic coefficient (i.e., $C_D$) as well as the stagnation point heat flux have been monitored during iterations. Solution convergence has been assumed for residuals dropped more than three orders of magnitude and when the aerodynamic coefficient and the stagnation point heat flux plots reached a constant value. So, convergence is assessed by matching both criteria. For example, Figure 19 shows the aerodynamic and stagnation point heat flux convergence histories for the R8 point 2D axis-symmetric simulation.

**Figure 19** Drag coefficient and stagnation point heat flux versus iterations for R8 2D axis-symmetric simulation

4.3 Numerical results

Example of 3D numerical analysis can be found in Figure 20 and Figure 21. Three-dimensional streamtraces, coloured by Mach number, past the capsule at $M_\infty = 3$ and $\alpha = 10$ deg and pressure distribution on the spacecraft front shield can be found in Figure 20.

**Figure 20** 3-D streamtraces past SRV, coloured by Mach, and pressure field on heatshield at $M_\infty = 3$ and $\alpha=10$ deg (Pezzella, Catalano and Bourgoing, 2012) (see online version for colours)
Figure 21 Static temperature field on SRV pitch plane and on two cross sections at $M_\infty = 22.07$ and $\alpha = 10$ deg (Pezzella, Catalano and Bourgoing, 2012) (see online version for colours)

This figure provides very interesting flowfield features as the flow expansion at capsule shoulder and the complexity of the base flow.

Figure 21 reports on the flowfield predicted around the SRV at $M_\infty = 22.07$ and 10 deg AoA. In particular, the figure shows the temperature contours provided on the capsule pitch plane and two flowfield cross sections.

The pressure distribution over the SRV forebody is also provided. As shown, the maximum flowfield temperature is close to about 7000 K since, due to the high Mach number, thermo-chemical processes occur behind the bow shock as species vibrational excitation and dissociation, as discussed hereinafter.

As shown in Figure 21, at hypersonic speed, the flowfield is dominated by a strong bow shock and is characterised by all the typical hypersonic flowfield features as shock waves very close to the body surface (i.e., thin shock layer), thick boundary layers, high temperatures and aerodynamic coefficients that can be nonlinear functions of angle of attack and etc. (Viviani and Pezzella; Viviani and Pezzella, 2010; Bertin, 1994; Anderson, 1989).

Indeed, the flow crossing the bow shock suddenly decelerates thus increasing the pressure and temperature in the shock layer close to the stagnation region. The high angle of cone after the rounded stagnation region (equal to 45 deg) causes the curvature of the shock wave and, therefore, the presence of an entropy layer that affects the results at the wall. Further, the strong shock wave causes a large sonic region, a smooth conical flow along the SRV conical part, and a strong flow expansion at the shoulder. This expansion, dominated by inviscid effects, has the effect of rapidly lowering the translational temperature, density and pressure of the gas, while the chemical state of the gas and the temperatures, that characterise the energy in the internal modes, tend to remain frozen
and the gas is still dissociated and excited. This aspect is very important by capsule aeroheating point of view. In fact, as the gas flows downstream, because recombination occurs slowly, the vibrational temperature of gas rises still higher with the consequence that the gas can radiate significantly in the afterbody region. In particular, at the rearward facing base of the SRV, the flow separates and creates a region of recirculating flow bounded by dividing streamlines. In fact, it is clearly shown the shear layer that starting from the SRV shoulder, converges on the capsule axis, undergoing normal shock at the neck of this flow to re-direct in the direction of the farfield. Hence, from neck flow it develops an oblique trailing shock wave, ultimately forming a viscous core or inner wake. Fluid in the inviscid wake crosses the trailing shock, increasing pressure, temperature and density thereby, and in continuing downstream this outer wake merges with the inner wake.

The O$_2$ and N$_2$ mass fractions contours past the SRV at M4 TP are shown in Figure 22 (e.g., axisymmetric 2D computations, whole domain).

**Figure 22** Species mass fractions contours field at M4 TP (see online version for colours)

At this flight conditions, the free stream Mach number is equal to 17.61. Therefore, the air crossing the bow shock is suddenly heated and converted in a reacting mixture. As one can see, at these flight conditions, the Oxygen completely dissociates while Nitrogen starts to dissociate.

Pressure and Mach number contours at M3 TP are shown in Figure 23, where the comparisons between ablated and not ablated aeroshape at M3 TP is reported. A shown, the shock layer is very narrow, as expected, and no differences apply between not ablated and ablated aeroshape, as also suggested by Figure 24.
Aerodynamic and aerothermodynamic analysis

Figure 23 Pressure and Mach number contours at M3 TP. Comparison between ablated and not ablated aeroshape (see online version for colours)

Figure 24 Cp and heat flux comparison between ablated and not ablated aeroshape at M3 TP without mass blowing effects

Figure 24 shows that centreline profiles of Cp and convective heat flux do not change significantly passing from not ablated to ablated aeroshape. Indeed, at the nose and along with the SRV conical part, the heat flux profile is quite the same for both aeroshapes. Difference arises only at the shoulder where the heat flux decreases in the case of ablated heat shield (larger local radius of curvature of shoulder).

Figure 25 shows the static pressure and Mach number front shield fields at M1 flight conditions.

As shown, a very strong shock wave is generated in front the capsule due to the very high Mach number (i.e., $M_\infty = 34.45$). As a consequence, a large contribution to the vehicle aeroheating comes from the radiative heating of the plasma flow within the shock layer. This can be taken into account for the SRV TPS design. Such a strong shock causes
molecules of atmosphere to be dissociated and ionised, and consequently the gas in the shock layer consists of molecules, atoms, ions and electrons. Temperature distributions along the capsule’s stagnation line when the SRV is flying at M1 trajectory point are given in Figure 26, in which translational and vibrational temperatures distributions are also highlighted.

**Figure 25** Static Pressure and Mach flowfields past SRV at M1 TP (see online version for colours)

![Figure 25](image)

**Figure 26** Temperatures and species mass fraction along with the stagnation line at M1 TP (see online version for colours)

![Figure 26](image)

As one can see, the temperature behind the bow shock is very high thus causing the complete Oxygen and Nitrogen dissociation inside the shock layer, as shown by the right side of Figure 26. Here, the distributions of the mass fraction of the neutral and ionised species along with the stagnation line are reported. This figure points out that N₂ and O₂ molecules dissociate rapidly and, successively, the generation of NO molecule, the atomic Nitrogen and Oxygen, and the ionisation of the molecules such as N₂, O₂ and NO, occur in the thermal non-equilibrium region.

In general, the level of each formed species reaches a state of near chemical equilibrium for a large portion of the shock layer. Then, at the edge of the boundary layer
the levels of the species start to change rapidly again as the temperature falls and the density rises through the boundary layer: N and O recombine to their molecular forms resulting in an increase in the levels of N₂ and O₂ and a fall in the levels of N and O. The reason for this is that the ionised species recombine with the electrons to form neutral species of N, O, N₂ and O₂ and NO, as shown in Figure 27.

Figure 27 Species mass fractions along with the stagnation line at M1/R3 TP (see online version for colours)

In particular, Figure 26 and Figure 27 illustrate also that a large portion of the shock layer is in thermo-chemical equilibrium (i.e., the temperature profiles are quite flat until the boundary layer is reached) and only near the shock and in the boundary layer there is a departure from this state: the non-equilibrium region is just downstream the bow shock wave, and the size is of the order of the shock wave thickness. In fact, even if Tᵥ increases much more slowly since it is density dependent, the two temperatures (T,Tᵥ) are nearly equilibrated throughout the shock layer.

Moreover, the temperature trends exhibit a sharp discontinuity at the shock wave with a rapid decreasing behind the bow shock due to finite rate chemical reactions. For instance, the translational-rotational temperature, T, reaches the maximum value at about x=14 mm, while the vibrational-electronic temperature, Tᵥ, is still much lower than T. Tᵥ begins to be equilibrated around x = 10 mm and continues to be equilibrated until the surface. On the contrary, the thermal non-equilibrium is observed at the region adjacent to the equilibrium one; i.e., from x = 14 mm to 10 mm. The equilibrated temperature amounts up to about 11000 K. Concerning the ionic species, there are substantial levels of O⁺ and N⁺ while the mass fractions of N₂⁺, O₂⁺ and NO⁺ are very low. In fact crossing the shock, O₂ is rapidly and highly dissociated to form O and O⁺. N₂ is dissociated to form N and NO by recombination with O but a small fraction of NO is created. A large part of the atomic nitrogen produced by the dissociation of N₂ is ionised in N⁺. Figure 27 shows that the mass fractions of N⁺ and O⁺ amount up to around 0.085 and 0.015, respectively. Note that the ionisation process is very important considering that it has a non-negligible impact on radiative heat flux at high velocity flight conditions. The prediction of the forebody pressure distribution at each trajectory points is shown in Figure 28, where the translational temperature along with the capsule stagnation streamline is also reported (right side).
This figure (left side) illustrates that pressure distributions (and thus aerodynamic forces) are affected by the sonic character of the shock layer. When supersonic, the pressure distributions on the conical flank are flat, which is characteristic of conical flow, when the entire forebody shock layer is subsonic, the elliptic nature of that flow results in higher, more rounded distributions. Moreover, the effect of local entropy values is also clearly shown. Indeed, the entropy layer, due to the curvature of the bow shock, swallowing causes the decreasing of pressure, at the sphere cone junction, up to reach the quite constant value of pressure at the conical skirt.

Figure 28 (right side) points out also that the shock layer changes along the descent flight due to real gas phenomena, as expected. Note that, the maximum temperature of 30000 K is attained just downstream bow shock that takes place when the SRV is flying at $M_\infty = 41.54$ at 71.86 km altitude (i.e., point R1). In fact, at this trajectory point the capsule features the maximum internal energy (i.e., kinetic plus potential). This huge mechanical energy converts into thermal energy when flow crosses the bow shock. As far as SRV aeroheating (without accounting for heat shield ablation) is concerned, heat flux distributions at capsule front shield centreline for flight conditions ranging from R1 to R6 are shown in Figure 29. Results refer to convective and radiative heat fluxes at the wall. The latter one has been computed with PARADE with the density, the molar fractions and the two temperatures (translational and vibrational) of all the R trajectory points coming from CFD simulations. As shown, Figure 29 highlights that the maximum radiative heat flux is equal to about 6 MW/m² and is reached at the R3/M1 flight conditions (i.e., $H_\infty = 58.73$ km altitude and $M_\infty = 34.45$). The convective peak heating reaches 8.5 MW/m², and it is attained at R5/M2 trajectory point (i.e., $H_\infty = 52.05$ km altitude and $M_\infty = 29.12$). Anyway, the maximum total heat flux is equal to about 12.5 MW/m² and it arises when the SRV is flying at $H_\infty = 58.73$ km altitude and $M_\infty = 34.45$ (i.e., R3/M1 TP). Therefore, plasma radiation is an additional contribution to surface aeroheating that must be taken into account in designing the SRV TPS. In particular, the generic convective heat flux profile in Figure 29 highlights that, after the peak at the stagnation point, the heat flux decreases along with the surface as the boundary layer develops up to an inflection point that corresponds at the end of the spherical shape of the capsule. Hence, it continues to decrease along with the conical part with a different shape and, then, it increases near the shoulder due to the local small radius of curvature and to the expansion that causes a reduction of the boundary layer thickness.
The skin friction coefficient \( C_f \) over SRV centreline is summarised in Figure 30 for several TPs. As shown, the peak of \( C_f \) is confirmed at the shoulder where boundary layer is extremely thin due to flow rotation around it.

The discussion of the flowfield features given previously for the non-ablating case is largely appropriate also for the flowfield analysis with mass blowing. This task deals with 8 NS in 2D-axi configuration and aims to address the impact of the flowfield contamination by TPM chemical species coming from heat shield recession, expected on high-speed earth entry, on capsule aerothermal performances. The gas model to consider is air (32 species) in thermo-chemical non-equilibrium conditions. For example, the Mach number and static pressure fields past the SRV front shield at R10/B3 TP (i.e., \( H_\infty=57.07 \) km altitude and \( M_\infty=33.07 \) at which occurs the maximum stagnation point total mass blow rate) are recognised in Figure 31.

The boundary layer, however, is significantly different with respect to the non-ablating cases (Roberts, 1995). Indeed, the species produced by the blowing along the stagnation streamline at R1/B1 and R10/B3 TP are plotted in Figure 32.
Figure 30  Skin friction coefficient for several R trajectory points (see online version for colours)

Figure 31  Mach number and static pressure contours field at R10/B3 TP (see online version for colours)

Figure 32  Ablation products on the stagnation streamline at R1/B1 and R10/B3 TP (see online version for colours)
The level of all of air species fall as the wall is approached and the presence of the blown gases becomes more dominant. Moreover, it is evident from the profiles that some of the ablation products are undergoing chemical change and, it is interesting to note that although most of the ablation products are restricted to the boundary layer there is a significant level of some of these products, specifically C, at positions slightly beyond the edge of the boundary layer. The ablation products in the boundary layer mainly consist of H₂, C₂H, C₂H₂, CO, C and H (Roberts, 1995). In particular, the level of C₂ falls rapidly as the distance from the body increases, because it dissociates to form C and C, and also indirectly leads to the formation of CN and HCN. However, the rise in the level of C in the boundary layer is mainly attributable to the dissociation of CO. The relatively slow rate at which the level of H falls with increasing distance from the wall suggests that either H₂ or alternatively some of the hydrocarbons present are dissociating (Roberts, 1995). Figure 33 and Figure 34 highlight C₂, C₃ and C, CO mass fractions field past the SRV, respectively.

**Figure 33** C₂ and C₃ mass fractions at R10/B3 TP (see online version for colours)

**Figure 34** C and CO mass fractions at R10/B3 TP (see online version for colours)
Note that the determination of the distribution of the products of the ablation process within the shock layer is very important since species such as $C_2$, $C_3$ and also CO have strong radiative properties. In particular, $C_2$ and $C_3$ possess absorption properties while CO is a strong radiator. Finally, the comparison of the convective heat flux at SRV centreline for all B trajectory points is shown in Figure 35. As one can see, heatshield ablation mitigates the convective heat fluxes the capsule has to withstand during descent, as expected. For instance, the maximum convective heat flux lowers from 8.5 MW/m$^2$ to 4 MW/m$^2$ when ablation is turned-on in the numerical CFD computations. Heatshield ablation reduces the surface gradients of temperature and that of various species mass fractions, causing a decrease of convective and diffusive heat-fluxes (Pezzella, Catalano and Bourgoing, 2012). Furthermore, CO, one of the main ablation products (for carbon based materials), lowers significantly the wall enthalpy.

**Figure 35** Convective heat flux at SRV centreline for all B TPs (see online version for colours)

On the other hand, the flowfield contamination by chemical species coming from heat shield recession is expected also to provide a blocking effect also on radiative heat flux and increase ionisation (design issues not addressed in the present research effort). As far as capsule aerodynamics is concerned, a summary review of SRV lift ($C_L$), drag ($C_D$), and pitching moment ($C_M$) coefficients (only drag in CFD analysis) that are calculated considering $S_{ref} = 0.95$ m$^2$ (i.e., maximum SRV cross section area), $L_{ref} = 1.1$ m and pole at centre-of-gravity. Figure 36 shows a typical mesh surface of SRV that has been used for the engineering-level computations, in which Surface Impact Methods (SIM) typical of hypersonics, such as Prandtl-Meyer expansion flow and modified Newtonian theories are widely applied (Viviani and Pezzella, 2012). Analysis has been performed also in FMF conditions. As an example of engineering level results, the static pressure distributions over the wetted vehicle surface for $M_\infty = 25$ and for two different angles of attack: 0 and 5 deg are summarised in Figure 37.

As shown, when the angle of attack ($AoA$) $\alpha$ increases the surface pressure distribution changes thus increasing on the capsule windside, as expected. At the same time flow expansion on the capsule leeside determines locally lower pressure contours. The lift, drag and pitching moment coefficients for $AoA$ ranging from 0 to 180 deg, in free molecular (at $M_\infty = 10.3$) and hypersonic continuum (at $M_\infty = 25$) flow conditions,
Aerodynamic and aerothermodynamic analysis

are summarised in Figure 38 and Figure 39, respectively. As one can see, capsule aerodynamic features a non-linear behaviour, typical of high-speed flow regime. In particular, lift coefficient nulls at zero deg AoA, as expected being SRV a symmetric aeroshape. In rarefied flow regime, it is essentially negligible for $0 < \alpha < 180$ deg; while in continuum flow $C_L$ reaches 0.2 at $\alpha = 60$ deg. Therefore, a lifting re-entry can be envisaged for this kind of mission, in contrast with classical ballistic flight, provided that the aeroshape can fly at AoA in static stability condition. On the other hand, the maximum drag coefficient in both regimes is of order one and it is attained at $\alpha = 0$ and 180 deg. Of course, the latter AoA does not work considering the TPS layout and capsule static instability at this angle. Trim analysis of SRV in free molecular flow, highlights that the capsule features four trim angles of attack (i.e., where the pitching moment at the centre of gravity is equal to zero, see green curve in Figure 38), namely $\alpha = 0$, 65, 112 and 180 deg.

Figure 36  SRV panel mesh

![SRV panel mesh](image)

Figure 37  Pressure coefficient at $M_\infty=25$ and $\text{AoA}=0$ and 5 deg (see online version for colours)

![Pressure coefficient](image)

However, the sign of $C_{Ma}$ points out that only $\alpha = 0$ and 112 deg are static stability pitch trim AoA. As illustrated in Figure 39, the chosen aeroshape in continuum hypersonic flow has two trim angles of attack (i.e., $C_M = 0$ at $\text{AoA} = 0$ deg and 180 deg).

But, for the 180 deg AoA, the slope of the pitching coefficient ($C_{Ma}$) is positive and consequently the aeroshape has only one single stable position for $\text{AoA} = 0$ deg. This trim AoA ensures that the probe will not flip over and depart from the expected re-entry in hypersonic regime. SRV drag profile in transitional flow conditions is summarised in Figure 40.
As one can see, the drag profile versus Knudsen features the classical expected s-shape and, ranging from rarefied to continuum flow conditions, points out that $C_D$ passes from about 2 to 1.
Comparison among present CFD results and literature data for $C_D$ in continuum flow is shown in Figure 41 (Mitcheltree et al., 1998).

**Figure 41** Drag coefficient of SRV comparison among present and literature results (Mitcheltree et al., 1998) (see online version for colours)

It presents an approximation of the drag coefficient across the Mach range for the 45-degree half-angle shape. The hypersonic value, 1.07, was computed at Mach 31.8 and 21.5 using the Langley Aerothermodynamic Upwind Relaxation Algorithm (LAURA) (Mitcheltree et al., 1998). The subsonic value, 0.65, comes from tests conducted in the Langley 20-foot Vertical Spin Tunnel (Mitcheltree et al., 1998). The supersonic and transonic values are from Brooks and Nichols wind tunnel data on a similar geometry. For the purposes of this feasibility study, the important data are the hypersonic value, which affects the heat pulse, and the subsonic value that determines the impact velocity (Mitcheltree et al., 1998; Pezzella et al., 2009). The drag coefficients of the present evaluation compare rather well with literature data, especially at very high Mach number flow conditions. Below Mach 7, forebody-only CFD will not accurately predict the aerodynamics. The calculations must include the afterbody and wake. Therefore, differences are recognised at rather low Mach number. In these cases, the effect of base drag could come from the differences between the back shell configurations between SRV and the capsule considered in the literature data shown in Figure 42 (Mitcheltree et al., 1998).

**Figure 42** SRV configuration comparison (Mitcheltree et al., 1998)
Numerical results provided so far refer to uncoupled simulations both for plasma radiation and heatshield ablation. As a result, an error affects these results. The estimation of such a discrepancy, however, cannot be estimated straightforward, provided that one should compare CFD results with and without coupling effects for each simulation. Anyway, enhanced computational effort flowfield analyses (i.e., fully coupled calculation) point out that the effects of radiation and ablation result in an averaged lower temperature field about the re-entry capsule with respect to the case of uncoupled simulations (Reynier, Marieu and Marraffa, 1999; Mazoué et al., 2005). This means that the shock layer in the coupled computations is narrower than the uncoupled ones. Thus, plasma radiation and heatshield ablation result in an enhanced non-equilibrium flowfield past the capsule. Moreover, the heat flux on the capsule is lower in the case of coupled simulations. Finally, from aerodynamic point of view enhanced non-equilibrium effects (i.e., real gas effects) and ablation (i.e., aeroshape modification and enhanced roughness) result in a variation of capsule aerodynamic performance (i.e., increase of drag coefficient) that, in turn, influence its descent trajectory and the aerothermal loading environment the capsule has to withstand during descent, as well.

5 Concluding remarks

In this work, numerical simulations have been carried out for a super-orbital re-entry capsule. A design analysis was performed to determine both aerodynamic and aerothermodynamic performances of an entirely passive earth entry capsule for sample return mission. Present design result refers to both numerical and engineering-based analysis of a 1.1 m diameter spherically blunted 45-deg half-angle forebody with a low-density ablative heat shield. Engineering-based design has been applied to assess flowfield regime and capsule aerodynamics and static stability in rarefied and continuum flow conditions; while thermo-chemical non-equilibrium Navier-Stokes simulations have been performed to evaluate flowfield past the re-entry vehicle with the aim to assess the effects of aeroshape modification, flowfield radiation, and of surface mass blowing on the capsule aerodynamics and aerothermodynamics. Numerical results comparisons highlight that centreline profiles of pressure coefficient and convective heat flux do not change significantly passing from not ablated to ablated aeroshape, thus suggesting that no differences in capsule aerodynamics and aerothermodynamics are expected. In particular, the convective peak heating reaches about 8.5 MW/m² when the capsule is flying at 52.05 km altitude and Mach 29.12.

Furthermore, design analysis points out that the plasma radiation is an additional contribution to surface aeroheating that must be taken into account in designing the capsule TPS. For instance, the maximum radiative heat flux is equal to about 6 MW/m² and is reached at 58.73 km altitude and Mach 34.45; this value sums to that of convective heating for a maximum total heat flux of about 12.5 MW/m² at this flight conditions.

On the contrary, the impact of the flowfield contamination by chemical species coming from heat shield recession, expected on high-speed earth entry, on capsule aerothermal performances is significant. Heatshield ablation reduces the surface gradients of temperature and that of various species mass fractions, causing a decrease of convective and diffusive heat fluxes. Furthermore, CO, one of the main ablation products (for carbon based materials), lowers significantly the wall enthalpy. Indeed, the convective heat flux lowers from 8.5 MW/m² to 4 MW/m² when ablation is turned-on in
the numerical CFD computations. As far as capsule aerodynamics is concerned, numerical simulations highlight that capsule aerodynamic features a non-linear behaviour, typical of high-speed flow regime. In particular, lift coefficient in rarefied flow regime is essentially negligible for $0 < \alpha < 180$ deg; while in continuum flow it reaches 0.2 at $\alpha=60$ deg. Therefore, a lifting re-entry can be envisaged also for this mission, in contrast with classical ballistic flight, provided that the aeroshape can fly at angle of attack in static stability condition.

On the other hand, the maximum drag coefficient in rarefied and continuum regimes is of order one and it is attained at $\alpha = 0$ and 180 deg. Of course, the latter angle of attack does not work considering the heatshield layout and capsule static instability at this angle. Trim analysis of capsule in free molecular flow, highlights that it features four trim angles of attack, namely $\alpha = 0, 65, 112$ and 180 deg. However, the slope of pitching moment curve points out that only $\alpha = 0$ and 112 deg are static stability pitch trim angle of attack. In continuum hypersonic flow, the aeroshape has two trim angles of attack. But, for the 180 deg, the slope of the pitching coefficient is positive and consequently the capsule has only one single stable position for 0 deg angle of attack. In between rarefied and continuum flow, the drag passes from about 2 to 1. Finally, this work is useful to assess the aerodynamic performances and stability along the entry trajectory path, and to elaborate and validate an engineering tool that couples: aeroshape aerodynamic; trajectory and stability; aerothermal environments and heat shield ablation.

References


