DEBRIS AERODYNAMIC INTERACTION AND ITS EFFECT ON RE-ENTRY RISK ASSESSMENT

Y. Prevereaud(1), J.-L. Verant(2), J.-M. Moschetta(3), and F. Sourgen(4)

(1) ONERA - The French Aerospace Lab, 31 055 Toulouse, France, Email: Ysolde.Prevereaud@onera.fr
(2) ONERA - The French Aerospace Lab, 31 055 Toulouse, France, Email: Jean-Luc.Verant@onera.fr
(3) ISAE - Institut Supérieur de l’Aéronautique et de l’Espace, 31 055 Toulouse, France, Email: jean-marc.moschetta@isae.fr
(4) ONERA - The French Aerospace Lab, 31 055 Toulouse, France, Email: Frederic.Sourgen@onera.fr

ABSTRACT

Aerodynamic interactions of objects in a continuum hypersonic and supersonic flow are numerically and analytically investigated for perfect and thermochemical equilibrium gas conditions. An innovative semi-analytical methodology has been developed and associated to the Laurence equations [4] in order to compute the aerodynamics coefficients of a sphere located downstream the shock wave issued from a primary object. The influence of the aerodynamic interactions on the trajectory of a debris cloud during atmospheric re-entry has also been assessed and compared to the trajectory of the same non-interacting objects. Finally, the influence of fragments interactions on heat flux and debris survivability during the re-entry has been investigated.

Key words: Space debris; Interaction; Aerothermodynamics; Atmospheric re-entry; Risk.

1. INTRODUCTION AND CONTEXT

SINCE 1957 and the orbital performance of the soviet satellite Spoutnik-1, the human activity in space has generated a great number of space debris. The increase of fragments produced by in-orbit collisions, combined to the hard atmospheric drag in low Earth orbits and polar orbits, could lead to an increasing number of uncontrolled atmospheric re-entries for the next decades. Indeed, over the last forty years, about 16,000 tons of these objects ranging from ten microns to several meters have done an atmospheric re-entry. Between 10 to 40 percent of that mass are estimated to have reached the ground [1]. Occasionally, debris having a projected area superior or equal to 100 $m^2$ and weighting more than 10 tons could also perform atmospheric re-entry. That kind of objects can be classified as highly hazardous with regard to the great number of fragments produced during re-entry which are able to survive to the critical aerothermodynamic environment. Therefore, they represent a potential threat to ground safety.

Several space agencies and research institutes have developed tools to predict the atmospheric re-entry of space debris and to assess their total casualty area, impact location as well as the state in which the elements may reach the ground. At the present time, according to public literature, seven major atmospheric re-entry codes have been developed through the world. They can be divided into two categories: object-oriented codes and spacecraft-oriented codes.

Object-oriented codes only analyze individually the satellite elements performing re-entry. In other words, this kind of code assumes that, at a given altitude, usually set between 75 to 85 kilometers, the satellite is broken down into its elementary parts and scattered. Then, these ones are followed independently during re-entry phase (there is no physical model of fragmentation, the break-up altitude is empirical). Therefore, the atmospheric re-entry analysis of a whole satellite is replaced within the individual analysis of its most critical parts. The six object oriented codes known are : DAS (NASA), ORSAT (NASA), ORSAT-J (JAXA), DRAMA/SESAM (ESA), Debrisk (CNES) and DRAPS (China).

The spacecraft-oriented codes, whose SCARAB (ESA-HTG Germany) is currently the unique code known, consist of modeling the complete satellite, as close as possible to the real one. The fragmentation and/or ablation of the spacecraft model depends on the aerothermodynamic environment encountered during the atmospheric re-entry. After the fragmentation has occurred, each element is individually analyzed.

However, the great number of fragments actually generated during re-entry may evolve independently, or, on the contrary, interact one another for a while. In addition, a fragment, moving into the wake generated by the main object, is submitted to very different aerothermodynamic conditions as compared to those located upstream. Therefore, one or more objects located in the wake may have a different trajectory and life time than if they were facing the upstream flow. This complex situation, commonly called aerodynamic interactions or flying formation pro-
cess, is not currently taken into account by existing atmospheric re-entry debris codes [2].

Nevertheless, in the literature several authors have experimentally, numerically and analytically investigated the aerodynamics interactions into a given fragments cloud. In particular, Schultz and Sugita [3] (1994) have studied the evolution (the dispersal and the deceleration) of a debris cloud during atmospheric re-entry. Experimental investigations performed at the NASA-Ames Vertical Gun as well as theoretical considerations have allowed them to identify the objects sizes (the dimensionless shock distance), the lateral velocity as well as the atmospheric density which are required to experiment the so called collimation effect [3]. Authors have pointed out the importance of that kind of fragmentation process for the phenomenon to occur.

In terms of numerical simulations, Passey and Melosh [9] (1980) investigated the reasons for cross-range dispersion of fragments in crater fields. They concluded that and spinning of the primary fragment. Artemieva and Shuvalov [7], [10] (1996 and 2001) performed numerical simulations of the flow around interacting fragments for different shapes. They computed drag and lift coefficients acting on bodies placed in the primary shock region as a function of their relative positions. In further simulations [10] the authors investigated the repulsive coefficient as a function of the distance between two fragments placed side by side. They coupled their three dimensional hydrodynamic code with a fragmentation model in order to follow the evolution of a few number of fragments. More recently, using CFD, Barri [8] (2010) computed the aerodynamic coefficients acting on a sphere located at different positions in perfect gas conditions in the supersonic regime. She identified the critical values for fragment sizes and positions for the collimation effect can occur.

Finally, the original work of Vashchenkov, Kashovsky and Ivanov [11] (2003) should be mentioned since they have investigated forces acting on cylinders placed into the shock region generated by a primary one in the transitional regime ($Kn \sim 0.02$). In the simulations performed using a DSMC method, a very large distance of wake behind the body ($\sim 25 \times d_1$) has been under consideration. Authors have concluded that smaller fragments ($d_2 \leq 0.001 \times d_1$) cannot experience the non-uniformity of the flow. Furthermore, they have pointed out that the secondary fragments could have an influence on the primary one according to the fragment and the subsonic region size.

In anyway, the space debris re-entry trajectory and the survivability issue require the use of codes having short response time due to a necessary sensitivity approach for generation of thousands of trajectories. Therefore, the coupling of a trajectory code with a CFD code, as performed by Artemieva and Shuvalov [7], [10], would be much too expensive with regard to the CPU cost when applied to the whole trajectory computation. Moreover, even the supercomputers could not follow a great number of fragments if interactions are accounted for. In that case, model reduction strategy must be considered.

Laurence [4] (2007) has developed an analytical method, based on Sedov [5] Blast Wave Analogy (BWA), in order to determine the drag and lift coefficients exerting on an infinite cylinder or a sphere located within the primary shock region. The analytical results have been compared to numerical simulations and measurements. A reasonable agreement has been found for most of locations studied for the secondary fragments. Therefore, a number of problems are intrinsic to Sedov BWA. Yet, the BWA is known to underestimate the shock radius as shown in Fig. 2. Furthermore, because of the high Mach number assumption, Sedov-Laurence’s method can be used in high hypersonic regime only. According to the author, the blast wave method is in good agreement with numerical results for high speed flows which Mach number is of the order of 50 and higher, which remains an asymptotic case. However, in the case of debris entering the Earth atmosphere, the maximum re-entry velocity value is around $M_\infty \sim 30$. Furthermore, in Sedov-Laurence approach, the flow is assumed to be a perfect gas, which is absolutely not representative of the reality of flows encountered during the atmospheric re-entry. Finally, the blast wave methodology is not valid anymore neither in the wake region located immediately behind the body (for $x/d_1 < 1.5$, according to Laurence) nor far downstream.

The present work aims at developing a new methodology, called semi-analytical methodology, taking into account the real gas effect at thermochemical equilibrium, which would be valid in the whole front fragment shock region, and which would have a short response time in order to quickly compute the debris trajectories. This new methodology will replace the BWA in the Laurence approach. Therefore, in this paper, the interactions between fragments in the hypersonic and supersonic regimes at perfect and thermochemical equilibrium gas conditions are investigated and discussed. Then the influence of fragments interactions on trajectory and debris life time is studied.

2. METHODOLOGY

2.1. Principles of the semi-analytical method

The present semi-analytical method consists of extrapolating, for an other trajectory point, the aerothermodynamics data issued from a numerical reference field ($P_{\infty,simu}$, $T_{\infty,simu}$, $V_{z,simu}$, $V_{r,simu}$). The reference field might be computed by a two dimensional axisymmetric finite volume method (Fig. 1), for instance.

Knowing the free stream conditions encountered at each flight point ($P_{\infty,simu}$, $T_{\infty,simu}$, $V_{\infty}$), the first object size (diameter $= d_1$), the relative position of the secondary fragment with the primary one $(x, r)$ and the numerical reference field conditions - as first object size, $d_1$, free stream conditions ($P_{\infty,simu}$, $T_{\infty,simu}$, $V_{\infty,simu}$),...
aerothermodynamics data for each point \((x_{\text{simu}}, r_{\text{simu}})\) of the reference simulation, then aerothermodynamics flow data \((P, T, V_x, V_y)\) encountered by the secondary object can be assessed.

Any change in freestream conditions during the atmospheric re-entry would lead to a modification of the flow structure. In particular, the increase or decrease of the upstream Mach number value leads to a compression or an expansion of the shock wave, respectively. Therefore, the first step consists of determining where to select the aerothermodynamics data in the numerical reference flow field in order to develop a formulation for the transformation \((x, r) \rightarrow (x_{\text{simu}}, r_{\text{simu}})\). In the present paper, it has been chosen to keep \(x\) unchanged \((x = x_{\text{simu}})\), and to ensure the conservation of flow rate in the wake.

The shock wave plays a key role in the geometrical transformations used in the present method so that its position must be predicted analytically the most accurately as possible. A new relation based on the one obtained by Billig [12] for the vertex radius curvature has been proposed.

\[
R_s = a \times \tan \left( \arcsin \left( \frac{1}{M_\infty} \right) \right) \sqrt{\left( \frac{x + a}{a} \right)^2 - 1}
\]

(1)

With

\[
a = \frac{b}{2} \left( \frac{c}{M_\infty - 1} \right)^d \frac{1}{\tan \left( \arcsin \left( \frac{1}{M_\infty} \right) \right)^2}
\]

(2)

The coefficient \(b\) is a function depending only on the Mach number. Eq. 1 is applicable for a perfect gas flow on a large Mach number range from Mach 2 to Mach 9.5. For a gas at thermochemical equilibrium flow, Eq. 1 is valid from Mach 9.5 to Mach 30.
2.2. The numerical reference flowfields

The numerical reference flowfields have been performed using the ONERA multi-physics platform CEDRE coupled with the CHARME solver for hypersonic gas flow. A two-dimensional axisymmetric hexa grid of 310 000 nodes has been used. 370 cells are equally distributed in the radial direction around the sphere (Fig. 4). All simulations have been performed on a parallel cluster, and a CFL number equal to 0.5 has been chosen for each of them. The Euler equations have been solved at first spatial order for an ideal gas ($\gamma = 1.4$) and a real gas at thermochemical equilibrium using the flux vector splitting scheme of Van Leer, associated to a Van Leer limiter. The time integration has been set to a one step implicit finite-volume approach.

![Figure 4. 2D axisymmetric mesh used for the reference field computation assuming gas at thermochemical equilibrium.](image)

The ballistic re-entry of a single sphere, representing a hollow $Ti_{eq}Al_{4}V$ tank of one meter diameter, has been simulated in order to identify flight points of interest. Computation has been performed using a 4th-order Runge Kutta scheme and a fixed time step. The hollow sphere enters the atmosphere at $7700 \text{ m/s}$, at an altitude of $78 \text{ km}$ and an entry flight path angle of $-1^\circ$. The semi-analytical methodology has been tested on some identified flight points. Two chosen points are presented on Fig. 5 and 6.

![Figure 5. Comparison at fixed $x = 1\text{ m}$ (from sphere center) of a numerical simulation ($V_\infty = 7400 \text{ m/s}$, $H = 50 \text{ km}$) with the results extrapolated using the semi-analytical method for a real gas from a numerical reference field computed for $V_\infty = 7634 \text{ m/s}$, $P_\infty = 101325 \text{ Pa}$ and $T_\infty = 300 \text{ K}$. Pressure (green), Temperature (blue), and density (red).](image)

In Fig. 5, the results, issued from the semi-analytical method (using a numerical reference field computed for $V_\infty = 7634 \text{ m/s}$, $P_\infty = 101325 \text{ Pa}$ and $T_\infty = 300 \text{ K}$), are compared with the numerical simulation of the targeted field: the flow around a one meter diameter sphere flying at $7400 \text{ m/s}$ and at an altitude of $50 \text{ km}$ ($P_\infty = 79.78 \text{ Pa}$ and $T_\infty = 270.65 \text{ K}$). Pressure, temperature, and density values are compared at station $x = 1 \text{ m}$ (center to center), $r$ varying from the axis ($r = 0$) to the shock wave location ($R_s$). Notice that these results could not be compared with results obtained using the BWA which is based on perfect gas assumption. Fig. 6 displays comparison between numerical and semi-analytical results in conditions of lower velocity ($V_\infty = 2023 \text{ m/s}$) and lower altitude ($H = 20 \text{ km}$, i.e. $P_\infty = 5530 \text{ Pa}$ and $T_\infty = 216 \text{ K}$), in perfect gas. In fig. 5 and 6, after the passing across the shock, the aerothermodynamics parameters achieved their freestream values again.

The error produced by the analytical shock equation being smaller, a good agreement between numerical and semi-analytical results can be observed on Fig. 5 and 6, even in the shock region. Indeed, the semi-analytical method is in part based on a geometrical modification of the shock as a function of Mach number variations during the atmospheric re-entry. Therefore, the uncertainty given by the semi-analytical methodology is
Lift \( (C_L) \) and drag \( (C_D) \) coefficients, calculated with analytical (Sedov BWA) and present semi-analytical (Prevendau method) methods on a secondary sphere, are directly compared on Fig. 8, for perfect gas conditions. AMR Euler numerical simulations performed by Laurence [4] (with AMROC solver) for \( M_\infty = 10 \) and \( H = 0 \text{ km} \) are used as reference to estimate the level of deviation with analytical/semi-analytical results. Besides, Fig. 9 compares present method with ONERA Navier-Stokes numerical computations (performed thanks to elsA solver) for \( M_\infty = 22.4 \) and \( H = 50 \text{ km} \), assuming thermochemical equilibrium gas. In the two cases presented here, the diameter ratio between the primary and secondary fragments, \( d_2/d_1 \), are respectively equal to 2 and 4, and various center-to-center lateral and axial displacements are proposed.

The fragment is supposed to be large enough relatively to the shock thickness in order to experience the non-uniformity of the flow. Therefore, a non-zero lift coefficient can be observed on the secondary sphere.

To determine aerodynamics coefficients for the modified Newtonian profile, the equations of lift \( (C_L) \) and drag \( (C_D) \) coefficients, proposed by Laurence [4], are used. Moreover, the modifications in aerodynamics coefficients equations proposed by Laurence [4] to take into account the shock-shock interactions, i.e. for \( R_s - d_2/2 \leq r \leq R_s + d_2/2 \), have been considered here. However, the higher the ratio \( d_2/d_1 \) is and the less correct the modification proposed by Laurence are (Fig. 7).

In Fig. 8 a clear improvement is observed with the semi-analytical method. As can be seen in Fig. 9 numerical and semi-analytical results are close. Moreover, in Fig. 8 and Fig. 9, a small peak can be mentioned with computational and semi-analytical results for \( r/R_s = 0.48 \) and \( r = 0.25 \text{ m} \) respectively. This is caused by the flow unsteadiness produced by interactions of the secondary sphere with the wake region or the separation shock created by the primary one. Contrarily to the BWA, this effect is present in the semi-analytical method through the numerical reference field, and seems to be well reproduced for other flight points. Nearby the shock, for \( r/R_s \) close to 1, lift and drag coefficients significantly increase, due to the impingement of the primary shock wave on the secondary sphere (Fig. 10). As a matter of fact, the distribution of pressure is significantly and obviously affected by shock-shock interactions. Once again, a clear improvement can be observed with the semi-analytical
on the atmospheric re-entry of an hydrazine tank. Indeed, in their paper, the authors concluded that the tank considered should survived to atmospheric re-entry. In order to study the fragment interactions, the survivability of the primary fragment must be ensured. The secondary fragment is supposed to embody a hollow $Ti_6Al_4V$ tank of 0.5 meter diameter and 2 mm wall thickness.

The sphere trajectories are performed thanks to the integration of 3-Dof equations of motion describing changes due to the applied forces. Integration is performed, as long as the two bodies have not reached the ground, thanks to a 4th-order Runge-Kutta scheme with a fixed time step. The initial entry velocity of the main fragment is assumed to be in the x-z plane, where the z axis is normal to the Earth surface. The Earth is supposed to be spherical and without rotation. The flow around the bodies behaves as real or perfect gas according to the flight conditions. The two bodies are supposed not to rotate and have no mass loss during the atmospheric re-entry. The atmospheric constant values are taken from tables of the US-76 model with linear interpolation between values.

The primary object is one meter diameter sphere representing a tank of 2 mm wall thickness, made in $Ti_6Al_4V$. The four-fifth of the tank are supposed to contain frozen hydrazine ($N_2H_4$) whose density is found equal to 1025 kg/m$^3$. So, the tank contain 422.4 kg of frozen $N_2H_4$. The objective was to keep the same conditions than the ones proposed in the study of Kelley and Rochelle [14].

Figure 11. Relative motion of the secondary fragment versus the primary one during the atmospheric reentry

Fig. 11 shows the relative motion of the secondary fragment compared to the primary one. As considered here, the secondary sphere stays in interaction 1.7% of the total re-entry time with the primary one. The consequence on ground dispersion is about 20 km, when fragment interactions are considered or not (Fig. 12).

5. ATMOSPHERIC RE-ENTRY ANALYSIS : INFLUENCE ON DEBRIS SURVIVABILITY

In this chapter, the influence of fragments interaction on the debris survivability is investigated.

$$\Phi_{\text{stored}} = \Phi_{\text{conv}} + \Phi_{\text{rad,g}} - \Phi_{\text{rad,w}} + \Phi_{\text{cond}}$$

(3)

The heat flux balance is computed only at the stagnation point of the spheres.
The conductive heat flux is given by:

$$\Phi_{cond} = k \nabla T + \Phi_{diff}$$

(7)

where $\Phi_{diff}$ is the diffusive heat flux of species. $\Phi_{diff} = 0$ for a non-catalytic wall and $\Phi_{diff} \neq 0$ for a catalytic wall.

In the first approach proposed here, the conductive heat flux at the stagnation point of the secondary sphere is supposed to be equal to zero. Indeed, because of the very thin wall thickness of the secondary hollow sphere, the temperature inside the sphere is supposed to reach the temperature outside almost instantaneously. Moreover, the wall is assumed to be non-catalytic.

Fig. 14 to Fig. 17 display for different flight points the wall temperature ($T_w$) and total heat flux at the stagnation point of the secondary sphere ($\Phi_{tot} = \Phi_{conv} + \Phi_{rad,g} - \Phi_{rad,w}$), when the fragment interactions are (solid line) or not (dashed line) considered. The results are given from the axis ($r/R_s = 0$) to the shock ($r/R_s = 1$), for $\frac{d_1}{d_2} = 2$ and $\frac{x}{d_1} = 0.75$. The computations have been realized thanks to the present semi-analytical method for a real gas at thermochemical equilibrium. Wall temperature is obtained by assuming radiative equilibrium. Once melting point is reached, the temperature is set at this value and the convective and radiative heat fluxes are then computed. The melting point chosen for the computations is the titanium oxide one ($T_{melt} = 2130 \text{ K}$). In fact, during the atmospheric re-entry, oxidation will appear almost instantaneously recovering the tank wall. Oxidation affects thermo-mechanical properties of the material and so could have an impact on the fragment survivability.

![Figure 14. Comparison of the wall temperature and total heat flux on the secondary sphere at the stagnation point when fragment interactions are considered (solid line) or not (dashed line), for the following freestream conditions: $V_\infty = 7700 \text{ m/s}, H = 78 \text{ km}$.

Considering the evolution at the stagnation point of the secondary fragment wall temperature in the shock layer, two observations can be done from Fig. 14 to Fig. 17:

First, the protection conditions of the bow shock layer depends of the flight point conditions. Indeed, at the flight point $H = 50 \text{ km}, V_\infty = 7400 \text{ m/s}$ (Fig. 16) the melting temperature is reached whatever the fragment position along the $r$ axis for $x/d_1 = 0.75$. So, the fragment
is not protected by the primary one. Inversely, in Fig. 17 the melting temperature is never reached, meaning that the fragment remains completely protected at the flight point $H = 20\ km$ and $V_\infty = 2023\ m/s$.

Secondly, the protection conditions of the bow shock layer depends of the fragment position. In fact, as Fig. 14 and 15 shows, the melting temperature is not reached everywhere in the shock layer. From present simulations, the protection conditions of the bow shock are flight dependent and also fragment position dependent.

REFERENCES