ELABORATION OF A NEW SPACECRAFT-ORIENTED TOOL: PAMPERO

Julien Annaloro\(^{(1)}\), Pierre Omaly\(^{(1)}\), Vincent Rivola\(^{(2)}\), Martin Spel\(^{(2)}\)

(1) CNES Toulouse, 18 avenue Edouard Belin, 31400 Toulouse, France
Email: julien.annaloro@cnes.fr; pierre.omaly@cnes.fr

(2) R.Tech, Parc Technologique Delta Sud, 09340 Verniolle, France
Email: vincent.rivola@rtech-engineering.com; martin.spel@rtech.fr

ABSTRACT
At present, CNES has developed two aerothermodynamics tools named DEBRISK and PAMPERO, in order to assess the ground risk of debris. This paper will be focused on PAMPERO, a new tool based on a spacecraft-oriented approach. This code calculates the entire trajectory of one object by 6 DOF simulations, coupled with a 3D thermal module and ablation. The different modules will be presented. Comparisons and validations have been performed.

1. INTRODUCTION
In the frame of the French Space Act concerning the space operations (LOS) voted in 2008, CNES is particularly interested by re-entries of space debris. Especially on the assessment of risks for every mission which has to be launched and operated from the French territory and in case of:

- A launch failure,
- An atmospheric re-entry of a satellite.

Thus, the maximum allowable probability to have at least one victim has been set at:

- \(2 \times 10^{-5}\) for a controlled re-entry,
- \(10^{-4}\) for an uncontrolled re-entry.

Accordingly, the prediction of the space debris survivability during the entry and the assessment of the prospective risk on ground, require the development of a complete multidisciplinary tool. At present, aerothermodynamics tools can be classified into two categories: the object-oriented tools and the spacecraft-oriented tools.

- The main idea of the object-oriented approach is to simplify the vehicle geometry from the break-up altitude into individual simple shapes, defined by the user (sphere, cylinder, box …). During the trajectory calculation, aerothermodynamics parameters of the object are time-dependent. Usually the object temperature is considered as uniform. DAS, DEBRISK, DRAMA, ORSAT, ORSAT-J are, for example, tools using this approach. Finally, the main advantage of this method is the relatively short calculation time.
- The spacecraft-oriented tools aim to simulate as real as possible an entire spacecraft where a mesh is initially applied. This approach shall be more accurate than the previous one due to the local calculations of the different aerothermodynamics parameters (pressure coefficient, convective heat flux …). Nevertheless, due to the more complex modelling strategy, more physical modelling efforts and computational resources are necessary. The tool SCARAB from ESA can be classified as such oriented tools.

This paper is in the framework of the second category of tools presented above and focus on the elaboration of a new CNES spacecraft-oriented tool named PAMPERO. Therefore, this paper aims to present the physical modules currently implemented.

2. PHYSICAL MODEL

2.1. Kinematics assumptions
PAMPERO is a spacecraft-oriented tool where 6 DOF simulations can be performed. The trajectory and the attitude of the object are integrated with the classical Runge-Kutta method. Aerodynamic forces are only due to pressure effects, inertia moments are assumed constant.

2.2. Mesh
In order to perform calculations with PAMPERO, a tetrahedral mesh has to be generated (cf. Fig. 1). The mesh is initially read in order to recognize surface or volume mesh and their possible connections. This operation is repeated as long as the object is ablated.

View factor calculations are also performed for each time step. Thus, a cell is considered numerically active if it is facing the flow and is not hidden by other cells: mathematically, the normal vector \(\vec{n}\) of a cell has to be in an opposite direction to the upstream flow velocity \(\vec{u}_{\infty}\).
Due to the mathematical formulation on the convective heat flux (cf. 2.4), the local curvature radius has to be accurately estimated for each cell. This procedure can be relatively easy for simple shapes (cf. Fig 3.) but may be problematic for more complex shapes. Different techniques have been tested in order to obtain a good precision on results and a good computing time (cf. Fig 4).

At last, the material is assumed as homogeneous, each cell has the same volumic mass.

2.3. Aerodynamic assumptions

As previously stated, the aerodynamic forces are only due to the pressure effects:

\[
\overline{f_p} = \frac{1}{2} \rho \omega U^2 c_p \bar{n}_i
\]  (1)

where \( c_p \) is the local pressure coefficient. Friction forces are not yet considered.

By estimating the Knudsen number

\[
Kn = \frac{\lambda}{L}
\]  (2)

where \( \lambda \) is the mean free path and \( L \) a representative physical length scale, the local pressure coefficients are calculated for the 3 different regimes:

- The analytical law of Bird [1] is used for the rarefied regime,
- The well-known modified Newton law [2] for the continuum regime is used such as

\[
c_p = c_{p,stag} \cos^2 \theta
\]  (3)

where \( \theta \) is the angle of attack and \( c_{p,stag} \) the pressure coefficient at the stagnation point, defined as follows

\[
c_{p,stag} = \frac{p_{stag} - p_\infty}{0.5 \rho_\infty U_\infty^2}
\]  (4)

- A “bridging” function \( \Phi(Kn) \) is used for the transitional regime such as

\[
c_p = c_p^L + (c_p^L - c_p^C) \times \Phi(Kn)
\]  (5)

2.4. Convective heat flux

The approach concerning the local convective heat flux calculation is similar to the local pressure coefficient calculation:
\[ Q_{\text{conv}} = Q_{\text{conv,stag}} \cdot f(r(\theta), P(\theta)) \]  
(6)

where \( Q_{\text{conv,stag}} \) is the convective heat flux at the stagnation point, \( f(r(\theta), P(\theta)) \) is a mathematical function depending on the local curvature radius and the local pressure. \( Q_{\text{conv,stag}} \) is generally estimated by empirical laws or correlations deduced from CFD calculations. Thus, comparisons have been performed on the convective heat flux calculated by PAMPERO and the MISTRAL CFD code from R.Tech.

**Continuum regime:**

Fig. 5. shows us a comparison for a sphere. We can notice a good agreement on the local convective heat flux. Tab. 1 presents the discrepancy between PAMPERO and MISTRAL on the integrated convective heat flux: the agreement is very good.

![Figure 5. R = 0.5 m, M = 15, \( \alpha = 0^\circ \)](image)

**Table 1.** Discrepancy between PAMPERO and MISTRAL on the integrated convective heat flux for a sphere

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Discrepancy (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>0.39</td>
</tr>
<tr>
<td>15</td>
<td>2.08</td>
</tr>
<tr>
<td>20</td>
<td>0.21</td>
</tr>
</tbody>
</table>

Fig. 6, 7 and 8 present comparison for a cylinder. Concerning the local convective heat flux, the agreement is relatively good. Nevertheless, some discrepancies appear close to edges. These effects are essentially due to:

- The estimate of the curvature radius close to these regions,
- The approach of the pressure coefficient calculation:
  - \( C_p \) depends only on the angle of attack in PAMPERO,
  - Flow acceleration close to edges can be calculated by CFD, not by PAMPERO.

**Rarefied regime:**

Comparisons have been also performed with DSMCFoam in the rarefied region. Fig. 9 presents a comparison for cylinder at \( \vartheta = 25^\circ \) and \( \alpha = 90^\circ \): the agreement on the local convective heat flux is good. It is

![Figure 6. R = 0.5 m, L = 3 m, M = 9, \( \alpha = 0^\circ \)](image)

![Figure 7. R = 0.5 m, L = 3 m, M = 9, \( \alpha = 30^\circ \)](image)

![Figure 8. R = 0.5 m, L = 3 m, M = 9, \( \alpha = 90^\circ \)](image)

**Table 2.** Discrepancy between PAMPERO and MISTRAL on the integrated convective heat flux for a cylinder

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Discrepancy (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>16</td>
</tr>
<tr>
<td>15</td>
<td>13</td>
</tr>
<tr>
<td>20</td>
<td>14</td>
</tr>
</tbody>
</table>

...
interesting to notice that recirculation of the flow is not predicted by PAMPERO at the cavity (the cell are considered as hidden in this configuration).

Figure 9. $R = 0.5 \text{ m}$, $L = 3 \text{ m}$, $M = 25$, $\alpha = 90^\circ$

2.5. 3D thermal module

The finite volume is applied for the heat transfert equation and for each cell:

$$T_i(t + \Delta t) = T_i(t) + \Delta t \ast RHS_i(t)$$  \hspace{1cm} (7)

$$RHS_i(t) = \frac{1}{V_i \rho c_p} \sum_j [Q_{\text{diff},i,j} + Q_{\text{conv},j} + Q_{\text{rad},j}]$$  \hspace{1cm} (8)

- For a face connected to another tetrahedron and not visible by the flow

$$Q_{\text{diff},i,k} = k \cdot \nabla^2 T_i, \quad Q_{\text{conv},j} = 0, \quad Q_{\text{rad},j} = 0$$

- For a face not connected to another tetrahedron but visible by the flow

$$Q_{\text{diff},i,k} = 0, \quad Q_{\text{conv},j} = 6 \text{DoF module}, \quad Q_{\text{rad},j} = \sigma \cdot c \cdot (T_i^4 - T_{\text{inf}}^4) \cdot A_j$$

- For a face not connected to another tetrahedron and not visible by the flow

$$Q_{\text{diff},i,k} = 0, \quad Q_{\text{conv},j} = 0, \quad Q_{\text{rad},j} = 0$$

In order to validate the thermal module, comparisons with OpenFOAM have been performed: an external power is set and assumed constant and uniform during all the calculation. At a chosen time, the comparison is made. Two different test-cases have been studied:

**Sphere test-case (cf. Fig. 10):**

- $R = 0.5 \text{ m}$
- $k = 237 \text{ W/m/K}$

**Plate test-case (cf. Fig. 11):**

- $l = 2 \text{ m}$, $L = 0.5 \text{ m}$, $e = 0.1 \text{ m}$
- $k = 237 \text{ W/m/K}$
- $C_p = 903 \text{ J/kg/K}$
- $\rho = 2787 \text{ kg/m}^3$
- $\text{Pow} = 400 \text{ kW}$
- $t = 200 \text{ s}$

Fig. 10 and Fig. 11 show a very good agreement between OpenFOAM and PAMPERO.

2.6. Ablation

A preliminary ablation module has been implemented in PAMPERO: a cell is removed when its melting temperature is reached. Accordingly, the recognition between 2D/3D mesh and the estimate of curvature radius have to be upgraded (cf. Fig. 12).

Fig. 13 shows us a test-case for a hollow sphere and for a fixed trajectory point:
Solid properties:

- $R = 0.5$ m
- $k = 7$ W/m/K
- $C_p = 807.5$ J/kg/K
- $\rho = 4437$ kg/m$^3$
- $\varepsilon = 0.302$
- $T_{mel} = 1943$ K
- $m = 3.5$ kg

Aerodynamic conditions:

- $\rho_{in} = 0.00371$ kg/m$^3$
- $T_{in} = 256$ K
- $P_{in} = 272$ Pa
- $U_{in} = 2890$ m/s
- $0^\circ$ for the angle of attack
- Moments are not calculated

The sphere being fixed in the flow and taking into account the aerodynamic conditions, ablation can occur on the front. By the view factor calculation, the rear of the sphere can see in turns the flow and ablation also occurs.

To summarise:

- 6 DOF calculations can be performed,
- Aerodynamic forces are only due to pressure effects,
- The pressure coefficient can be calculated for the 3 regimes,
- The convective heat flux is generally estimated by empirical laws or correlations with CFD,
- A 3D thermal module has been implemented,
- A preliminary ablation module has been also implemented.

The different studies presented here show interesting agreements between CFD from MISTRAL and PAMPERO. Nevertheless, PAMPERO is still in development where improvements of some modules are necessary. More comparisons and validations have also to be done.

4. REFERENCES


3. CONCLUSION

PAMPERO is a new CNES spacecraft-oriented tool. Its development allows a better understanding of the various physical phenomena during the re-entry and to find new ways to improve the DEBRISK software.