Earth re-entry capsule CFD simulations taking into account surface roughness and mass injection at the wall

J.B. Vos*
CFS Engineering, EPFL Innovation Park, Batiment A, 1015 Lausanne, Switzerland
Email: jan.vos@cfse.ch
*Corresponding author

A. Bourgoing and J. Soler
Airbus DS, 51-61 Route de Verneuil, 78033 Les Mureaux, France
Email: alexis.bourgoing@airbus.com
Email: jacques.soler@airbus.com

B. Rey
CFS Engineering, EPFL Innovation Park, Batiment A, 1015 Lausanne, Switzerland
Email: benoit.rey@cfse.ch

Abstract: Future space exploration missions will likely use capsule type ballistic re-entry vehicles that will re-enter the earth atmosphere at very high speed (velocities in the order of 11.7 km/s or higher, this compared to 7.5 km/s for the US Space Shuttle). Passive concepts will be employed for the deceleration, and the thermal protection system (TPS) needs to be dimensioned to protect the vehicle and samples from the effects of the aerodynamic heating. The technology readiness level of these passive concepts is today still rather low, leading to large design margins. This paper is concerned with the design of a possible earth re-entry capsule, it discusses the re-entry trajectory and heat loads encountered during descent. Detailed CFD simulations were made for three selected points on the re-entry trajectory. The ablation of the TPS material was taken into account through a mass injection boundary condition. The turbulence models employed in the study were modified to account for surface roughness due to ablation. Calculated heat fluxes and shear stresses for a smooth wall are compared with results from empirical correlations and showed a fairly good agreement. For rough walls, the CFD results show a substantial increase in convective heat flux, while the injection of mass at the wall reduces the convective heat flux.

Keywords: aerothermodynamics; blowing; CFD; earth re-entry capsule; roughness modelling.

Reference to this paper should be made as follows: Vos, J.B., Bourgoing, A., Soler, J. and Rey, B. (2015) 'Earth re-entry capsule CFD simulations taking into account surface roughness and mass injection at the wall', Int. J. Aerodynamics, Vol. 5, No. 1, pp.1–33.
An important step for space exploration activities to acquire a more accurate knowledge of the earth, universe and environment is to develop the capability to send vehicles into space that select, collect and finally return samples from our solar system (including asteroids) to earth for further analysis.

To return these samples, very high-speed earth re-entry is today considered as one of the most promising technologies to make this kind of space exploration missions affordable from the cost point of view. The design of a high-speed earth re-entry sample return vehicle requires a strong technological base and relies on a good understanding of the environment encountered during the earth re-entry. A high-speed earth re-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 500 MJ/m²), where the radiative part is rather important.

The main objective of the EU funded FP7 project RASTAS SPEAR (Radiation-Shapes-Thermal Protection Investigations for High-Speed Earth Re-entry) is to increase Europe’s knowledge in high-speed re-entry vehicle technology to allow for planetary exploration.
missions in the coming decades. This is accomplished by the study and development of innovative, critical technologies that will lead to an increase in knowledge and in a reduction of uncertainties and margins used in the design of sample return vehicles.

One of the first candidates for a planetary exploration is the Mars Sample Return (MSR) mission. The re-entry technology is considered as one of the critical technologies required for the MSR mission, in particular from the point of view of planetary protection requirements. High-speed re-entry technology is considered as the most promising technique to make sample return missions affordable. The prime requirement is to ensure containment of the Mars samples during the intense Earth entry, descent and impact phase of the mission.

NASA has proposed a fully passive concept for a MSR vehicle (Mitcheltree et al., 2000; Mitcheltree and Kellas, 1999) similar to the ballistic concept shown in Figure 1. To ensure containment, the entry technology must either include sufficient redundancy of each critical sub-system or eliminate the need for that sub-system. Due to mass restrictions on the MSR, NASA considered it essential to keep a simple and robust concept, thus avoiding the addition of redundant systems. The final proposed concept is based on the two following choices:

- a direct and fully ballistic re-entry
- no use of active deceleration systems (parachutes for example), which means a concept solely based on aerodynamics for deceleration and attitude control, and on TPS materials to sustain the heat loads.

This fully passive concept is compliant with re-entry missions that are strongly demanding in terms of planetary protection requirements. The aerodynamic concept of such an entry vehicle shall ensure a preferential impact direction. From a pure aerodynamic point of view, a sphere containing the canister would be a very good solution, but from the architectural point of view, such an Earth return vehicle (ERV) will need some remaining equipment for attachment to the carrier and for canister integration in the ERV. This is why a sphere-cone forward shape is preferred. The samples, packed in a hardened container, are surrounded by sufficient energy-absorbing material (to limit loading during ground impact) and covered by a thermal protection system (TPS) adapted to the very high heat fluxes (>10 MW/m²) during re-entry (see Figure 1). The TPS is a key element of the ERV, and its function is to limit heating to the structure and payload. For an ERV type configuration, the forward heat shield will likely be a low density ablator. This means that the shape of the vehicle will change during re-entry, that the surface of the forward heat shield becomes rough, there is mass injection in the boundary layer, followed by burning of the TPS material in the boundary layer (note that the burning of the TPS material nor the influence of radiative heat transfer is considered in the study presented here).

Surface roughness will promote transition to turbulence in the boundary layer, and will significantly increase the turbulent convective surface heat transfer. This effect is known since many decades but is still poorly understood and difficult to predict in the high-speed/hypersonic flow regime. It is however believed that in the high-speed case, where severe surface recession can be expected, the surface blowing might mitigate the heat transfer increase due to surface roughness.
One of the activities in the RASTAS SPEAR project was to study the influence of roughness and blowing on the heating of a ballistic capsule type re-entry vehicle, and the results of these studies are presented here.

2 Capsule shape and re-entry trajectory

One of the first tasks of the RASTAS SPEAR project was to assess the environmental data for a high-speed entry with a focus on earth entry. During this task, an aeroshape was selected and a flight domain has been determined with classical constraints on several parameters (maximum heat flux, maximum heat load, maximum g-load and impact velocity). The resulting aeroshape is not based on complete trade-offs but on relevant solutions based on the experience of Airbus DS. The main features of the selected shape are summarised in Figure 2.

Two sizing trajectories were computed using the usual design criteria: the maximum heat flux trajectory (Steep trajectory) and the maximum heat load trajectory (Swallow trajectory). For the present work, the maximum sizing heat flux trajectory was selected for the CFD computations. The conditions at entry at the altitude of 120 km are summarised in Table 1.

Table 1  Conditions at entry

<table>
<thead>
<tr>
<th>Max heat flux trajectory (steep)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Entry velocity @ 120 km</td>
</tr>
<tr>
<td>Flight path angle</td>
</tr>
</tbody>
</table>
Figure 3 shows the typical evolutions in time of several essential aero-thermodynamic parameters during the re-entry.

**Figure 3** Heat flux, Mach, Z, V vs. time for the FPAe = -12.5° @ 120 km trajectory (see online version for colours)

### 3 CFD analysis

#### 3.1 Overview NSMB CFD code

The NSMB (Navier Stokes Multiblock) CFD code used in this work was initially developed in 1991 at the Swiss Federal Institute of Technology (EPFL) in Lausanne, Switzerland. From 1992 to the end of 2003, NSMB was further developed in the NSMB consortium (Vos et al., 1998, 2002), which included several universities (EPFL, Lausanne, Switzerland, SERAM, Paris, France, IMF-Toulouse, France, KTH, Stockholm, Sweden), one research establishment (CERFACS, Toulouse, France) and several industrial partners EADS-France (Airbus France and EADS Space Technologies), SAAB Military Aircraft and CFS Engineering. Since 2004, NSMB is further developed by EPF-Lausanne, ETH-Zürich, IMFS-Strasbourg, IMF-Toulouse, Technical University of München, University of the Army, München, Polytechnique Montreal, Airbus DS Space Technologies, CFS Engineering and RUAG Aviation.

NSMB employs the cell-centred finite volume method using multiblock structured grids to discretise the flow field. The patch grid and chimera grid approaches are available to facilitate the grid generation for complex geometries.

Various space discretisation schemes are available to approximate the inviscid fluxes, among them, the second and fourth order centred schemes with artificial dissipation, and second, third and fifth-order upwind schemes. Viscous fluxes are approximated on a staggered grid using a second order scheme.

The space discretisation leads to a system of ordinary differential equations, which can be integrated in time using either the explicit Runge-Kutta scheme or the
semi-implicit LU-SGS scheme. To accelerate the convergence to steady state, the following methods are available:

- local time stepping
- implicit residual smoothing (only with the Runge-Kutta scheme)
- multigrid and full multigrid (grid sequencing)
- pre-conditioning for low Mach number.

A large number of well tested turbulence models are available in NSMB, among them the algebraic Baldwin-Lomax turbulence model, the Spalart-Allmaras 1 equation turbulence model and different variants of the $k-\omega$ family of 2 equation turbulence models.

NSMB includes different levels of modelling to simulate hypersonic flows. Chemical equilibrium air is computed using polynomials (TGAS/VGAS) (Srinivasan, Tannehill and Weilnunster, 1987). Chemical non-equilibrium is available for air, N$_2$ and CO$_2$. For air, 5, 7, 11 and 13 species models have been implemented, and different reaction schemes are available to describe the chemistry (Park, Kunn-Dang, Gardiner). Thermo-chemical non-equilibrium is available for each molecule in air using the Landau-Teller equation for the translational-vibrational energy exchange with the relaxation time calculated using the semi-empirical Millikan and White formula.

The transport coefficients (viscosity, thermal conductivity and diffusion coefficients) are calculated using the Blottner or Gupta-Yos model for the viscosity, using the Eucken relation to calculate the thermal conductivity and Fick’s law for the diffusion coefficients.

Recently, the NSMB code was extended with the possibility to adapt the external grid boundary to the bow shock. This option removes the problem of the carbuncle introduced by the numerical scheme due to the non-alignment of the grid with the shock, and for a capsule geometry, it improves considerably the solution on the heat shield. Figure 4 shows an example of the shock adaptation for the chemical non-equilibrium calculation over the earth Re-entry Capsule at a free stream Mach number of 25.11. In this case, the grid was adapted two times during the time integration process, and one clearly sees the improvement of heat flux contours through the use of shock adaption. In addition the carbuncle, which is visible in the top figure of the Mach number contours, has been almost removed.

### 3.2 Roughness and blowing modelling

A review of surface roughness modelling can be found in Aupoix and Spalart (2003), Knopp, Eisfeld and Calvo (2009) and Aupoix (2007). The most commonly used method in CFD simulations is the so-called equivalent sand-grain approach. It consists of two steps: in the first step, the rough surface is reduced to an equivalent reference surface. In the second step, the turbulence model is modified to reproduce the effects of rough surfaces on the skin friction and heat transfer.

Popular turbulence models used in Aerospace are the Spalart-Allmaras turbulence model (Spalart and Allmaras, 1994) and the $k-\omega$ family of turbulence models (Wilcox, 1988) and for this reason they were selected for extension with a roughness model. For both models transport equation(s) are solved to model turbulent variables. These transport equations contain so-called production and destruction terms, which require the knowledge of the distance of a point to the nearest wall cell. It has been shown that for
rough walls this distance needs to be modified by the addition of an offset distance \( d_0 \) (see Knopp, Eisfeld and Calvo (2009) and Aupoix (2007) for a detailed discussion).

**Figure 4** Example of the use of shock adaptation for the earth Re-entry Capsule. Left: Mach number contours in the symmetry plane; Right: heat flux contours on the heat shield. Top row: non adapted grid, other rows results on successively adapted grids (see online version for colours).

### 3.2.1 Spalart-Allmaras turbulence model

In the Spalart-Allmaras turbulence model (Spalart and Allmaras, 1994), a transport equation is solved for the turbulent viscosity \( \tilde{\nu} \). This transport equation was constructed using empiricism, dimensional analysis, Galilean invariance and selected use of the molecular viscosity.
Both production and dissipation terms in the transport equation for $\tilde{\nu}$ require the knowledge on the distance $d$ of a grid point to the nearest wall. At smooth walls the value of $\tilde{\nu}$ equals zero.

Extension of the Spalart-Allmaras turbulence model for use with rough walls is straightforward (Aupoix and Spalart, 2003) and consists of the following two steps:

1. The variable $\tilde{\nu}$ at the wall is non-zero and needs to be calculated.
2. The wall distance $d$ needs to be modified by adding the so-called offset distance $d_0$.

Following Aupoix and Spalart (2003), the normalised value of $\tilde{\nu}$ at the wall can be computed from

\[
\tilde{\nu}' = \left(0.377 \ln \frac{h_s^*}{10} - 0.447\right) \exp\left(-\frac{h_s^*}{70}\right)
+ 1.25710 \times 10^2 h_s^* \left[1 - \exp\left(-\frac{h_s^*}{70}\right)\right]
+ \max\left(0, \ln \frac{h_s^*}{10}\right) \min\left[1, 1.36 \exp\left(-\frac{h_s^*}{250}\right), 25 \exp\left(-\frac{h_s^*}{100}\right)\right]
\]

with $h_s^*$, the normalised equivalent sand roughness height that is computed from

\[
h_s^* = \frac{h}{\nu_e}
\]

(2)

The variable $\nu$ is the kinematic viscosity and $u_e$ is the friction velocity at the wall. The normalised offset distance $d_0^*$ can be directly computed from the relation above:

\[
d_0^* = \frac{\nu_e}{\kappa}
\]

(3)

with $\kappa$ the Von Karmann constant.

3.2.2 $k$-$\omega$ family of turbulence models

For the $k$-$\omega$ family of turbulence models two transport equations are solved, one for the turbulent kinetic energy $k$, and the second one for the specific dissipation rate $\omega$ (Wilcox, 1988). The $k$-$\omega$ model suffers from its dependency of the value of $k$ in the free stream region, and Menter (1993) proposed a variant that acts like the $k$-$\omega$ model near the wall and like the $k$-$\varepsilon$ model in the free stream region. In addition, a shear stress transport correction term was added to this model. The resulting model is called the $k$-$\omega$ SST model and has become very popular for aerospace applications.

Two possibilities exist to extend the $k$-$\omega$ family of turbulence models for rough walls and both have been implemented in the NSMB code.
3.2.3 Method by Wilcox for rough walls

The original version of the $k-\omega$ turbulence model (Wilcox, 1988) included boundary conditions for rough walls and walls with blowing. In the case of a rough wall, the value of the dissipation rate $\omega$ at the wall is computed from

$$\omega_w = \frac{u}{U} S_R$$

with

$$S_R = \begin{cases} \left( \frac{50}{h^+} \right)^2 & \text{for } h^+ < 25 \\ \left( \frac{100}{h^+} \right)^2 & \text{for } h^+ \geq 25 \end{cases}$$

To permit the use of this method with the Menter Shear Stress variant of the $k-\omega$ turbulence model, the modification proposed by Hellsten (1998) for the turbulence viscosity in the near wall region was implemented in NSMB.

3.2.4 Method by Knopp, Eisfeld and Calvo (2009) for rough walls

Knopp, Eisfeld and Calvo (2009) proposed an extension of the $k-\omega$ turbulence models to account for wall roughness. This extension is based on the work of Aupoix and Spalart (2003) for the Spalart-Allmaras turbulence models. This extension can be summarised as follows:

1. The wall distance $d$ needs to be modified by adding the so-called offset distance $d_o$. The value of $d_o$ is computed as $0.03 h_s$ with $h_s$ the equivalent sand roughness height.

2. The value of the turbulent kinetic energy $k$ at a rough wall is computed from

$$k_{\text{rough}} = u_{\tau}^2 / \beta_k^{1/2}; \quad \phi_1 = \min \left( 1, \frac{h^+}{90} \right)$$

$u_\tau$ is the friction velocity at the wall and $\phi_1$ is a blending function.

3. The value of $\omega$ at a rough wall is computed from

$$\omega_{\text{rough}} = \frac{u_{\tau}}{\beta_\omega^{1/2} \kappa d_o}; \quad d_o = \phi_2 0.03 h_s$$

$$\phi_2 = \min \left[ 1, \left( \frac{h^+}{90} \right)^{2/3} \right] \min \left[ 1, \left( \frac{h^+}{45} \right)^{1/4} \right] \min \left[ 1, \left( \frac{h^+}{60} \right)^{1/4} \right]$$
To be consistent with the limit of $\omega$ near a hydrodynamic smooth wall, the value above is limited to the value of $\omega$ for a smooth wall using the boundary condition proposed by Menter (1993)

$$\omega_{wR} = \min\left(\frac{u_\tau}{B_\tau^{1.5} \kappa_d^6}, \frac{60 \nu}{\beta_u d^2}\right)$$ \hspace{1cm} (8)

with $d$ the distance to the nearest wall and $\beta_u$ a constant equal to 0.075.

### 3.2.5 Method of Wilcox (1988) for walls with blowing

Only the $k-\omega$ family of turbulence models has been modified to account for blowing at the wall by implementing the method proposed by Wilcox (1988). Similarly to rough walls, the value of $\omega$ at the wall is computed from

$$\omega_{wB} = \frac{u_\tau}{\nu} S_B$$ \hspace{1cm} (9)

with

$$S_B = \frac{20}{\nu^* \left(1 + 5 \nu^*_n\right)}.$$ \hspace{1cm} (10)

$\nu^*$ is the blowing velocity at the wall normalised with the friction velocity at the wall.

### 3.2.6 Extension of the method of Wilcox for walls with both roughness and blowing

Following a suggestion from Aupoix (2012), the value of $\omega$ at the wall in case of rough walls with blowing is computed from

$$\omega_{wRB} = \frac{\omega_{wR} \omega_{wB}}{\omega_{wR} + \omega_{wB}}$$ \hspace{1cm} (11)

### 3.3 Calculation matrix

Three trajectory points were selected for the computations presented in this paper. The computations are all made in the turbulent hypersonic regime and the selection of a transition criterion is therefore needed. The criterion used here is simply an estimate from the free-stream Reynolds number based on the probe diameter $D$. The critical value of $5.10^5$ was selected according to the ARD post-flight data (Tran, Paulat and Boukhobza, 2007).

$$\text{Re}_{\text{crit}} = \left(\frac{\rho_\infty u_\infty D}{\mu_\infty}\right) \geq 5.10^5$$
The selected computation conditions are compared to $Re_{\infty}, D, \text{total heat flux and altitude versus time, in Figure 5. For the present reference trajectory, the transition occurred at } Z = 51.9 \text{ km.}

**Figure 5** Selected calculation conditions on the reference trajectory (see online version for colours)

The three conditions for the CFD analysis are summarised in Table 2. The chemical non-equilibrium calculations were made using a 13 species air model considering 22 chemical reactions. The chemical species and their free stream mass fractions are given in Table 3 and the chemistry model and forward reaction rates are summarised in Table 4. This chemistry model was proposed in the RASTAS SPEAR project and is based on the 11 species chemistry model by Park (1993) with several modifications in the reaction rates suggested recently (Boyd et al.; Chernyi and Losev, 2003; Lee et al., 2007; Tsang and Herron, 1991). In addition, the ionisation of Argon (R11 in Table 4) (Gökçen) and the charge transfer reaction R16 (Park, 1993) were added.

According to the trajectory points selected for the present work, condition 1 can be considered as the limit case between thermal non-equilibrium and thermal equilibrium flow, which justifies the use of the thermal equilibrium assumption for this condition.

The chemical equilibrium calculations were made using a model based on polynomials that provide temperature, pressure, speed of sound and the transport coefficients as function of the density and internal energy (Srinivasan, Tannehill and Weilmunster, 1987).

The roughness heights given in Table 2 are translated into an equivalent sand roughness height required by the implemented roughness models. This was done using Dirling’s correlation (Dirling, 1973)
using a value of Λ of 8.64, see (Aupoix, 2012) for more details. This yielded equivalent sand roughness heights of 557 μm (rough 1) and 1154 μm (rough 2), respectively.

**Table 2** Calculation matrix

<table>
<thead>
<tr>
<th>Condition 1</th>
<th>Condition 2</th>
<th>Condition 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Altitude (km)</td>
<td>48.36</td>
<td>43.14</td>
</tr>
<tr>
<td>$V_\infty$ (m/s)</td>
<td>8280</td>
<td>5681</td>
</tr>
<tr>
<td>$p_\infty$ (Pa)</td>
<td>97.84</td>
<td>189.53</td>
</tr>
<tr>
<td>$ρ_\infty$ (kg/m$^3$)</td>
<td>1.2594 10$^{-3}$</td>
<td>2.5491 10$^{-3}$</td>
</tr>
<tr>
<td>$T_\infty$ (K)</td>
<td>270.65</td>
<td>259.02</td>
</tr>
<tr>
<td>Mach</td>
<td>25.11</td>
<td>17.61</td>
</tr>
<tr>
<td>$ReD_\infty \times 10^6$</td>
<td>0.673</td>
<td>0.968</td>
</tr>
<tr>
<td>$T_{wall}$ (K)</td>
<td>3000</td>
<td>3000</td>
</tr>
<tr>
<td>Roughness $h$ (μm)</td>
<td>250 and 500</td>
<td>250 and 500</td>
</tr>
<tr>
<td>Gas model</td>
<td>Chemical</td>
<td>Chemical</td>
</tr>
<tr>
<td>Chemical composition at wall</td>
<td>non-equilibrium</td>
<td>non-equilibrium</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>Spalart and k-ω</td>
<td>Spalart and k-ω</td>
</tr>
</tbody>
</table>

**Table 3** Chemical species and free stream mass fractions

<table>
<thead>
<tr>
<th>Species</th>
<th>Mass fraction</th>
</tr>
</thead>
<tbody>
<tr>
<td>N$_2$</td>
<td>0.75548</td>
</tr>
<tr>
<td>O$_2$</td>
<td>0.23161</td>
</tr>
<tr>
<td>Ar</td>
<td>0.01291</td>
</tr>
<tr>
<td>N</td>
<td>0.</td>
</tr>
<tr>
<td>NO</td>
<td>0.</td>
</tr>
<tr>
<td>O</td>
<td>0.</td>
</tr>
<tr>
<td>NO$^+$</td>
<td>0.</td>
</tr>
<tr>
<td>N$^+$</td>
<td>0.</td>
</tr>
<tr>
<td>O$^+$</td>
<td>0.</td>
</tr>
<tr>
<td>N$_2^+$</td>
<td>0.</td>
</tr>
<tr>
<td>O$_2^+$</td>
<td>0.</td>
</tr>
<tr>
<td>e$^-$</td>
<td>0.</td>
</tr>
<tr>
<td>Ar$^+$</td>
<td>0.</td>
</tr>
</tbody>
</table>
### Table 4  
Chemical model for 13 species air

<table>
<thead>
<tr>
<th>Reaction</th>
<th>Reaction Products</th>
<th>Reaction Antecedents</th>
<th>$K_r$ (cm$^3$/mole/s)</th>
<th>$n$</th>
<th>$T_a$ (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>R1</td>
<td>N$_2$ + Molecules = N + N + Molecules</td>
<td>N$_2$ + Atoms</td>
<td>7.00 $10^{21}$</td>
<td>–1.60</td>
<td>113200</td>
</tr>
<tr>
<td></td>
<td>e$^-$</td>
<td>e$^-$</td>
<td>3.00 $10^{22}$</td>
<td>–1.60</td>
<td>113200</td>
</tr>
<tr>
<td>R2</td>
<td>O$_2$ + Molecules = O + O + Molecules</td>
<td>O$_2$ + Atoms</td>
<td>2.00 $10^{23}$</td>
<td>–1.50</td>
<td>59500</td>
</tr>
<tr>
<td></td>
<td>Atoms</td>
<td>Atoms</td>
<td>1.00 $10^{22}$</td>
<td>–1.50</td>
<td>59500</td>
</tr>
<tr>
<td>R3</td>
<td>NO + Molecules = N + O + Molecules</td>
<td>NO + Atoms</td>
<td>9.64 $10^{14}$</td>
<td>0</td>
<td>74700</td>
</tr>
<tr>
<td></td>
<td>N$_2$</td>
<td>N$_2$</td>
<td>1.45 $10^{15}$</td>
<td>0</td>
<td>74700</td>
</tr>
<tr>
<td>R4</td>
<td>NO + O = O$_2$ + N</td>
<td>NO + O</td>
<td>8.40 $10^{12}$</td>
<td>0</td>
<td>19400</td>
</tr>
<tr>
<td>R5</td>
<td>N$_2$ + O = NO + N</td>
<td>N$_2$ + O</td>
<td>6.40 $10^{17}$</td>
<td>–1.00</td>
<td>38400</td>
</tr>
<tr>
<td>R6</td>
<td>N + N = N$_2$ + e$^-$</td>
<td>N + Atoms</td>
<td>2.00 $10^{13}$</td>
<td>0</td>
<td>67500</td>
</tr>
<tr>
<td>R7</td>
<td>O + O = O$_2$ + e$^-$</td>
<td>O + Atoms</td>
<td>1.10 $10^{13}$</td>
<td>0</td>
<td>80600</td>
</tr>
<tr>
<td>R8</td>
<td>N + O = NO$^+$ + e$^-$</td>
<td>N + O</td>
<td>5.30 $10^{12}$</td>
<td>0</td>
<td>31900</td>
</tr>
<tr>
<td>R9</td>
<td>N + e$^-$ = N$^+$ + e$^-$</td>
<td>N + e$^-$</td>
<td>2.50 $10^{14}$</td>
<td>–3.82</td>
<td>168200</td>
</tr>
<tr>
<td>R10</td>
<td>O + e$^-$ = O$^+$ + e$^-$</td>
<td>O + e$^-$</td>
<td>3.90 $10^{13}$</td>
<td>–3.78</td>
<td>158500</td>
</tr>
<tr>
<td>R11</td>
<td>Ar + e$^-$ = Ar$^+$ + e$^-$</td>
<td>Ar + e$^-$</td>
<td>2.50 $10^{14}$</td>
<td>–3.82</td>
<td>181700</td>
</tr>
<tr>
<td>R12</td>
<td>NO$^+$ + O$_2$ = O$_2^+$ + NO</td>
<td>NO$^+$ + O$_2$</td>
<td>2.40 $10^{13}$</td>
<td>0.41</td>
<td>32600</td>
</tr>
<tr>
<td>R13</td>
<td>O$_2^+$ + N$_2$ = N$_2^+$ + O$_2$</td>
<td>O$_2^+$ + N$_2$</td>
<td>9.90 $10^{12}$</td>
<td>0</td>
<td>40700</td>
</tr>
<tr>
<td>R14</td>
<td>O$_2^+$ + N = N$^+$ + O$_2$</td>
<td>O$_2^+$ + N</td>
<td>8.70 $10^{13}$</td>
<td>0.14</td>
<td>28600</td>
</tr>
<tr>
<td>R15</td>
<td>O$_2^+$ + O = O$^+$ + O$_2$</td>
<td>O$_2^+$ + O</td>
<td>4.00 $10^{12}$</td>
<td>–0.09</td>
<td>18000</td>
</tr>
<tr>
<td>R16</td>
<td>N$^+$ + N$_2$ = N$_2^+$ + N</td>
<td>N$^+$ + N$_2$</td>
<td>1.00 $10^{12}$</td>
<td>0.50</td>
<td>12200</td>
</tr>
<tr>
<td>R17</td>
<td>O$^+$ + N$_2$ = N$_2^+$ + O</td>
<td>O$^+$ + N$_2$</td>
<td>9.10 $10^{11}$</td>
<td>0.36</td>
<td>22800</td>
</tr>
<tr>
<td>R18</td>
<td>O$^+$ + NO = N$^+$ + O$_2$</td>
<td>O$^+$ + NO</td>
<td>1.40 $10^{5}$</td>
<td>1.90</td>
<td>15300</td>
</tr>
<tr>
<td>R19</td>
<td>NO$^+$ + N = N$_2^+$ + O</td>
<td>NO$^+$ + N</td>
<td>7.20 $10^{13}$</td>
<td>0</td>
<td>35500</td>
</tr>
<tr>
<td>R20</td>
<td>NO$^+$ + N = O$^+$ + N$_2$</td>
<td>NO$^+$ + N</td>
<td>3.40 $10^{13}$</td>
<td>–1.08</td>
<td>12800</td>
</tr>
<tr>
<td>R21</td>
<td>NO$^+$ + O = O$_2^+$ + N</td>
<td>NO$^+$ + O</td>
<td>7.20 $10^{12}$</td>
<td>0.29</td>
<td>48600</td>
</tr>
<tr>
<td>R22</td>
<td>NO$^+$ + O = N$^+$ + O$_2$</td>
<td>NO$^+$ + O</td>
<td>1.00 $10^{12}$</td>
<td>0.50</td>
<td>77200</td>
</tr>
</tbody>
</table>

The influence of the ablation of the TPS material was simulated through the injection of air (blowing) at the solid wall. The mass flow rates of the ablation products are determined using an in-house Airbus DS material response tool in one-dimensional mode. This tool determines the heat transfer, wall and in-depth temperature and density, recession surface and mass blow rate (total and for each injected species) of an elemental layout of materials stacked up constituting the TPS. The inputs needed are the local conditions (surface pressure, ...
convective and radiative heat fluxes and the elemental composition of the TPS). The surface pressures along the heat shield are determined using CFD. The convective and radiative heat fluxes are obtained from empirical correlations using the local similarity method (Anderson, 1989). For the convective heat flux, Fay-Riddell is used at the stagnation point, while Kemp, Rose and Detra (1995) are employed in the laminar regime and Vaglio-Laurin (1960) in the turbulent regime. In the latter case, a laminar flow is assumed until \( x = 0.04 \), followed by a turbulent flow downstream of this point. For the radiative heat fluxes, the Tauber and Sutton (1991) formulation is employed all along the profile of the heatshield.

The TPS material employed is a low density carbon phenolic ablator similar to PICA (Tran et al., 1996). The composition of the injected products (pyrolysis and ablation) at the surface is obtained by assuming that the surface is in chemical and thermal equilibrium locally (pressure and temperature). The energy balance at the wall permits to determine the mass flow rates due to ablation \((m_c)\) and to pyrolysis \((m_g)\) as follows:

\[
-\lambda \nabla T_s = q_{c,w} + q_{r,w} + \varepsilon \sigma (T^4_s - T^4_w) + \dot{m}_c \Delta H_{comb} + \dot{m}_g \Delta H_{abl}
\]  

The term \(-\lambda \nabla T_s\) is the heat flux into the material. The terms \(q_{c,w}\) (with blockage effects) and \(q_{r,w}\) refer to the convective and radiative heat fluxes coming from the shock layer. The term \(\varepsilon \sigma (T^4_s - T^4_w)\) refers to the radiative heat flux from the wall. \(\dot{m}_c \Delta H_{comb}\) is the heat flux due to reactions of pyrolysis gas and \(\dot{m}_g \Delta H_{abl}\) is the heat flux due to TPS material ablation. The pyrolysis is modelled using an Arrhenius law.

The blockage effect due to blowing is taken into account by a blowing rate correction expressed as follows:

\[
\Psi_c = \frac{\alpha}{\alpha_0} = 1 - \frac{\dot{m}_c}{\dot{m}_c} \eta_{pyr} - \frac{\dot{m}_c}{\dot{m}_c} \eta_{abl}
\]

In this equation, \(\Psi_c\) is the convective blockage factor, \(\alpha\) and \(\alpha_0\) are the convective heat transfer coefficients with and without blockage, and \(\eta_{pyr}\) and \(\eta_{abl}\) are the accommodation factors for pyrolysis and ablation and depend on the laminar-turbulent flow regime. Typical values of \(\eta\) vary between 0.4 (turbulent flow) and 0.6 (laminar flow) for moderate blowing (Reynier, 2013). The computations to determine the mass flow blowing rate were performed using a uniform TPS thickness of 56 mm on a grid having 55 points distributed along the profile as shown in Figure 6. As can be seen in this figure, the grid was refined on the sphere and in the cone/tore area.

For each condition in Table 2, the mass flow rate was provided in different points along the capsule geometry, see Figure 7. The NSMB CFD code was modified to permit the reading of the mass flow rate data files and to interpolate the data on the CFD mesh.

The chemical composition of air at the wall for the chemical non-equilibrium calculations was obtained using the fully catalytic wall assumption (the chemical composition at the wall is computed from the chemical equilibrium assumption at the given wall temperature and pressure). In terms of heat fluxes, the catalytic wall
assumption is closer to reality than a non-catalytic wall assumption. However, it can be expected that the computed heat fluxes are over estimated.

**Figure 6** Location along the heat shield profile of points selected for pyrolysis and ablation computations

**Figure 7** Mass flow rate along the body for the four conditions (see online version for colours)
Two turbulent models were used in the CFD simulations for all three conditions, the Spalart-Allmaras turbulence model (Spalart and Allmaras, 1994) with the Aupoix model (Aupoix and Spalart, 2003) to account for the roughness, see also Section 3.2 and the $k$-$\omega$ Menter Shear Stress turbulence model (Menter, 1993) with the Knopp model (Knopp, Eisfeld and Calvo, 2009) for the roughness. For condition 1, for the lowest roughness height (rough 1) also the $k$-$\omega$ Wilcox turbulence model was employed using both the Knopp and Wilcox roughness models (Wilcox, 1988). When using the $k$-$\omega$ turbulence models, the approach of Spalart and Rumsey (2007) for the free stream values of $k$ and $\omega$ was employed. In this approach, the free stream value of the turbulent kinetic energy is set to a low value and additional source terms are added to the equations for $k$ and $\omega$ to sustain the turbulence. For the calculations discussed in this report, the free stream value of $k$ was set to $10^{-5}$.

3.4 Numerical parameters

All calculations were made on a 3D grid, considering $\frac{1}{4}$ of the geometry. The structured grid was generated to permit the use of shock adaptation. Figure 8 shows the initial grid in the symmetry plane and the block topology. An O-grid was used to cluster grid points near solid walls, and 81 grid points were placed in this grid. The total number of grid points was 4.5 Million cells. The first grid point near the wall was placed such that the $y^+$ value was well below 1. Depending on the turbulence model and for a smooth wall, the wall value of $y^+$ varied between 0.05 and 0.5.

Figure 8 Initial grid in the symmetry plane and block topology (see online version for colours)

All calculations were performed using the second order central space discretisation scheme using artificial dissipation. A blend of the Jameson artificial dissipation and a TVD scheme was used (Swanson and Turkel, 1992). The equations were integrated in time using the LU-SGS semi-implicit scheme (Yoon and Jameson, 1986). Local time stepping was employed to accelerate the convergence to steady state. The initial CFL number was set to 0.1, which was increased to 100 during the time integration for the calculations for conditions 1 and 2, and increased to $10^9$ for the calculations for condition 3.

Shock adaptation was employed either at steps 2000 and 3000, or steps 3000 and 4000. If needed, a third shock adaptation was made to remove any wiggles in the shock
shape. For each calculation, around 8000 time steps were made for conditions 1 and 2, and 6000 time steps for condition 3. Convergence was judged by the convergence of the L2-residual of the continuity equation normalised by the value of this residual at the first step, the convergence of the aerodynamic coefficients and by comparing the heat flux on the body saved at 1000 steps interval. Figure 9 shows typical figures of the L2-residual and the drag coefficient $C_D$ (starting at step 1000 for visualisation reasons) for two different calculations. One can clearly observe the jump in residues and $C_D$ when adapting the shock, but convergence is recovered in about 200 steps.

Figure 9 Typical L2-residual (left) and CD convergence (right) curves (see online version for colours)

3.5 Empirical correlations

The computed results in Section 3.6 are compared with results from the empirical correlations used to compute the blowing rate. For a laminar flow, the method of Kemp, Rose and Detra (1995) is used, while for a turbulent flow the work of Vaglio-Laurin (1960) is employed. In this case, a laminar flow is assumed until $x = 0.04$ m ($y = 0.12$ m), followed by a turbulent flow downstream of this point. In case of blowing, two models are used for the convective blockage factor $\Psi_c$, see Equation (14). In model 1, $\Psi_c$ is computed from

$$\Psi_c = 1 - \eta B_0' = 1 - \eta \frac{\dot{m}}{\alpha_0}$$

with $\dot{m}$ the blowing rate and $\alpha_0$ the convective heat transfer coefficient without blowing. In this model, the constant $\eta$ is set to 0.51 for turbulent flows.

In model 2, the convective blockage factor is calculated from an approach based on a one-dimensional Couette flow approximation (Mickley et al., 1954).

$$\Psi_c = \frac{2\lambda B_0'}{e^{2\lambda B_0'} - 1}$$

with $\lambda$ an accommodation factor (Moyer and Rindal, 1968) set to 0.4 in the turbulent regime.
Roughness effects are taken into account using the formulation of Powars (Wool, 1975) in which the convective heat transfer coefficient for a smooth wall is multiplied by a factor $K_r$:

$$a_e = \alpha_e K_r$$ (17)

The factor $K_r$ is a function of the parameter $\xi$ defined as

$$\xi = \frac{\sqrt{\text{Re}_k}}{\sqrt{\text{St}_t} \left( \frac{T_s}{T_p} \right)^{1/3}} \text{ with } \text{Re}_k = \frac{\varphi u_t k_t}{\mu_e}$$ (18)

$k_t$ is the roughness height, $\text{St}_t$ the Stanton number for a smooth wall and the subscript $e$ indicates quantities at the edge of the boundary layer. According the value of $\xi$, $K_r$ takes the following values:

$$K_r = \begin{cases} 1 & \text{if } \xi \leq 10 \\ 1 + \frac{2}{3} \log_{10} (\xi) - 1 & \text{if } 10 < \xi < 10^4 \\ 3 & \text{if } \xi \geq 10^4 \end{cases}$$ (19)

Radiative heat transfer is not included in the heat fluxes obtained from these correlations.

3.6 Discussion of the calculated results

3.6.1 Results condition 1

Figures 10 to 13 show the computed heat flux along the wall using different viscous modelling assumptions (laminar flow, turbulent flow using, respectively, the Spalart-Allmaras, $k-\omega$ Menter Shear Stress and $k-\omega$ Wilcox turbulence models), for a smooth wall and for rough walls using the two different roughness heights, and with and without blowing at the wall. The results are compared with heat fluxes obtained from empirical correlations for a laminar and turbulent flow provided by Airbus DS, see also Section 3.5.

- Smooth wall without blowing.

Figure 10 shows the computed heat flux for a smooth wall without blowing. The results for a laminar flow are very close to the empirical correlations. Differences occur just downstream of the sphere-cone junction (located at $y = 0.2$ m) until $y = 0.3$ m, where heat fluxes from the empirical correlations are slightly higher than the CFD results.

In turbulent regime, both CFD results and correlations are scattered. We can divide the CFD models into two families. CFD models which predict a laminar-turbulent transition on the profile (Spalart-Allmaras and $k-\omega$ Wilcox) and those which predict transition near the stagnation point at the nose ($k-\omega$ Menter Shear Stress). The heat fluxes computed using the Spalart-Allmaras turbulence model coincide with the laminar results until the sphere-cone junction which triggers a slow transition to a turbulent flow. A similar observation can be made for the heat fluxes computed using the $k-\omega$ Wilcox
turbulence model with a transition to a turbulent flow further downstream ($y = 0.3$ m), and a faster transition process as is reflected in the heat flux levels. The heat fluxes computed using the $k-\omega$ Menter Shear Stress turbulence model show a weakly turbulent flow (ratio of turbulent to laminar viscosity around 2) at the nose until the sphere-cone junction, followed by a fully turbulent flow downstream of this point. Due to this behaviour, the computed heat fluxes using the $k-\omega$ Menter Shear Stress turbulence model are the highest. The heat fluxes calculated using the empirical correlations follow the laminar results until $y = 0.12$ m, followed by a turbulent flow behaviour downstream of this point. Of the three turbulent models employed, the $k-\omega$ Menter Shear Stress turbulence model predicts the highest heat fluxes that at the tore of the capsule ($y = 0.55$ m) are more than twice as high as the heat fluxes computed from the empirical correlations. The heat fluxes calculated using the other two turbulence models remain below the result from the empirical correlations until the flow is fully turbulent. At the tore of the capsule, these two turbulence models predict heat fluxes that are up to 50% higher than computed from the empirical correlation.

**Figure 10** Computed heat flux along the wall, condition 1, smooth wall, no blowing (see online version for colours)

- **Blowing effects on smooth wall**

Figure 11 shows the computed heat fluxes for a smooth wall with blowing together with results from empirical correlations. As discussed in Section 3.5, two models are used for the empirical correlations.
The blowing effect reduces the heat fluxes in both laminar and turbulent regimes as expected. The computed heat fluxes for a laminar flow are slightly lower than the empirical correlations until the sphere-cone junction; differences are larger downstream of this point. As for the case with no blowing, the computed heat fluxes using the Spalart-Allmaras turbulence model follow the laminar flow results until the sphere-cone junction, followed by a transition to a turbulent flow. The results obtained using the $k-\omega$ Menter Shear Stress turbulence model show a weakly turbulent flow in the nose region followed by fully turbulent flow. Of the two empirical correlations, model 2 predicts slightly higher heat fluxes. At the tore maximum heat fluxes are obtained using the $k-\omega$ Menter Shear Stress turbulence model (about twice as high as the empirical correlations), followed by the Spalart-Allmaras model (about 50% higher than the empirical results). Comparing Figure 10 and 11 permits to assess the influence of blowing on the computed heat fluxes and the results are summarised in Table 5.

<table>
<thead>
<tr>
<th></th>
<th>Mid-sphere ($y = 0.1 \text{ m}$)</th>
<th>Sphere-Cone Junction ($y = 0.2 \text{ m}$)</th>
<th>Mid-cone ($y = 0.37 \text{ m}$)</th>
<th>Tore</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laminar NSMB</td>
<td>$-15%$</td>
<td>$-32%$</td>
<td>$-38%$</td>
<td>$-27%$</td>
</tr>
<tr>
<td>Laminar Correlation</td>
<td>$-13%$</td>
<td>$-14%$</td>
<td>$-14%$</td>
<td>$-14%$</td>
</tr>
<tr>
<td>Spalart-Allmaras</td>
<td>$-17%$</td>
<td>$-25%$</td>
<td>$-6%$</td>
<td>$-2%$</td>
</tr>
<tr>
<td>$k-\omega$ Menter Shear Stress model</td>
<td>$-14%$</td>
<td>$-15%$</td>
<td>$-11%$</td>
<td>$-10%$</td>
</tr>
<tr>
<td>Correlation Model 1</td>
<td>$-14%$</td>
<td>$-15%$</td>
<td>$-16%$</td>
<td>$-15%$</td>
</tr>
<tr>
<td>Correlation Model 2</td>
<td>$-11%$</td>
<td>$-12%$</td>
<td>$-12%$</td>
<td>$-12%$</td>
</tr>
</tbody>
</table>
As can be seen from this table, and as expected, blowing leads to a 10 to 20% reduction in heat flux except for the Spalart-Allmaras turbulence model at the mid-cone and tore points. For a laminar flow, NSMB predicts a higher reduction in heat flux compared to the correlations for a laminar flow. Figure 11 shows that the curve for the heat flux computed using the laminar correlation is parallel to the NSMB result, but with an offset towards a higher heat flux (for the case without blowing the NSMB results and laminar correlations were in good agreement, see also Figure 10). As a result, the reduction in heat flux calculated using the laminar correlation is lower.

The heat flux calculated using the Spalart-Allmaras turbulence model follows the laminar results until the sphere-cone junction. After this point, the heat flux is increasing more rapidly compared to the case without blowing, see also Figure 10. Blowing favours transition to turbulence so the flow is more turbulent for the case with blowing, reducing the effect of blowing on the heat flux when compared to the no blowing case.

- Rough wall without blowing.

Only the Spalart-Allmaras turbulence model was used to compute the heat fluxes for rough walls without blowing, and the results are shown in Figure 12 together with the results from empirical correlations and results for a laminar flow. For both roughness heights, the heat fluxes computed using the Spalart-Allmaras turbulence model follow the laminar result (for a smooth wall) until the sphere-cone junction where transition occurs.

Figure 12 Computed heat flux along the wall, condition 1, rough wall, no blowing (see online version for colours)
Downstream of this point the heat fluxes are increasing and more rapidly for the largest roughness height. Heat fluxes computed using the empirical correlations for a turbulent flow follow the laminar results at the nose until the imposed transition point. Again the results computed for the largest roughness height yields the highest heat fluxes. Table 6 summarises the increase in heat flux due to a rough wall compared to a smooth wall at four points. For the Spalart-Allmaras turbulence model this increase is small at the first two points because the flow is laminar until the sphere-cone junction. At the third location, the increase in heat flux is substantial, in particular for the largest roughness height. The correlations show an increase in heat flux of around 20% for the lowest roughness height and about 40% for the largest.

Table 6 Influence of roughness on computed heat fluxes

<table>
<thead>
<tr>
<th></th>
<th>Mid-sphere (y = 0.1 m)</th>
<th>Sphere-Cone Junction (y = 0.2 m)</th>
<th>Mid-cone (y = 0.37 m)</th>
<th>Tore</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spalart-Allmaras model rough 1 no blowing</td>
<td>+5%</td>
<td>+2%</td>
<td>+25%</td>
<td>+27%</td>
</tr>
<tr>
<td>Spalart-Allmaras model rough 2 no blowing</td>
<td>+5%</td>
<td>+2%</td>
<td>+71%</td>
<td>+54%</td>
</tr>
<tr>
<td>Correlation rough 1 no blowing</td>
<td>+21%</td>
<td>+21%</td>
<td>+20%</td>
<td>+22%</td>
</tr>
<tr>
<td>Correlation rough 2 no blowing</td>
<td>+39%</td>
<td>+42%</td>
<td>+41%</td>
<td>+43%</td>
</tr>
</tbody>
</table>

- Rough wall with blowing.

Figure 13 shows the computed heat fluxes for the rough walls (with two roughness heights) including blowing at the wall, and Table 7 summarises the change in heat flux at the four selected points compared to the corresponding smooth case without blowing.

For the Spalart-Allmaras turbulence model, the flow remains laminar until the sphere-cone junction, which explains the decrease in heat flux which corresponds to the decrease in heat flux due to blowing. Both blowing and roughness promote transition to a turbulent flow which is reflected in the large increase in heat flux at the mid-cone point. The results for the $k$-$\omega$ Menter Shear Stress model show an increase in heat flux that is slightly higher at the mid-cone location. This can be attributed to a higher turbulence level in the flow due to the combined effects of blowing and roughness. The correlations all show an increase in heat flux, with the results for blowing model 2 slightly higher compared to model 1.

Figure 14 shows the ratio of turbulent to laminar viscosity for the 2 rough wall cases with blowing using both the Spalart-Allmaras and $k$-$\omega$ Menter Shear Stress turbulence models. As can be seen in this figure, the turbulent viscosity is below 0.1 in the nose region for the results obtained using the Spalart-Allmaras turbulence model, confirming the laminar flow in this region. For both turbulence models regions of high turbulent viscosity are larger for the rough 2 cases.
Earth Re-entry Capsule CFD simulations

Figure 13 Computed heat flux along the wall, condition 1, rough wall with blowing (see online version for colours)

Table 7 Influence of the combined effect of blowing and roughness on the computed heat fluxes

<table>
<thead>
<tr>
<th>Model/Merit Rough 1 Blowing</th>
<th>Mid-sphere (y = 0.1 m)</th>
<th>Sphere-Cone Junction (y = 0.2 m)</th>
<th>Mid-cone (y = 0.37 m)</th>
<th>Tore</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spalart-Allmaras model</td>
<td>-17%</td>
<td>-24%</td>
<td>+34%</td>
<td>+26%</td>
</tr>
<tr>
<td>rough 1 blowing</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spalart-Allmaras model</td>
<td>-17%</td>
<td>-24%</td>
<td>+90%</td>
<td>+51%</td>
</tr>
<tr>
<td>rough 2 blowing</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>k-ω Menter Shear Stress</td>
<td>+10%</td>
<td>+6%</td>
<td>+6%</td>
<td>+8%</td>
</tr>
<tr>
<td>model rough 1 blowing</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>k-ω Menter Shear Stress</td>
<td>+17%</td>
<td>+13%</td>
<td>+11%</td>
<td>+11%</td>
</tr>
<tr>
<td>model rough 2 blowing</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Correlation rough 1</td>
<td>+3%</td>
<td>+2%</td>
<td>+1%</td>
<td>+3%</td>
</tr>
<tr>
<td>blowing model 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Correlation rough 1</td>
<td>+7%</td>
<td>+7%</td>
<td>+6%</td>
<td>+8%</td>
</tr>
<tr>
<td>blowing model 2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Correlation rough 2</td>
<td>+19%</td>
<td>+20%</td>
<td>+19%</td>
<td>+21%</td>
</tr>
<tr>
<td>blowing model 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Correlation rough 2</td>
<td>+24%</td>
<td>+26%</td>
<td>+24%</td>
<td>+26%</td>
</tr>
<tr>
<td>blowing model 2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 15 shows the computed wall shear stress for the different calculations made. In the nose region the results obtained using the $k$-$ω$ Menter Shear Stress turbulence model predicts the highest wall shear stress due to the fact that this model predicts a weakly turbulent flow in this region. The computed wall shear stress using the Spalart-Allmaras turbulence model follows the laminar result with blowing until transition to turbulence takes place at the sphere-cone junction. Again, these results correspond until this point closely to the empirical results. Once the flow is fully turbulent, one can observe that the
wall shear stress increases with increasing roughness height, and that the blowing at the wall is not sufficient to compensate this increase.

**Figure 14** Ratio turbulent to laminar viscosity for the calculations for condition 1. The grey colour indicates regions with values below 0.1, the black colour values above 20 (see online version for colours)

Figure 16 shows the Mach number and Temperature contours. Maximum temperatures in the flow reach around 14'700 K just downstream of the shock in the nose region. Due to the shock adaptation, the free stream boundary is close to the body.

Figure 17 shows the computed heat flux along the wall using the $k-\omega$ Wilcox turbulence model for the cases with and without blowing, and using the lowest roughness height. Two different approaches for the roughness modelling were used, the original approach proposed by Wilcox (1988) and the method proposed by Knopp, Eisfeld and Calvo (2009). The heat flux obtained for a laminar calculation is added to the figure as
reference. From Figure 17, it can be seen that the flow remains laminar until \( y = 0.25 \text{ m} \), and transition to turbulence occurs downstream of this point. The precise location depends on the use of blowing and of the roughness model. Both surface roughness and blowing at the wall promote transition to a turbulent flow. Comparing the two implemented roughness modelling approaches shows that the original Wilcox model leads to a faster transition to turbulence compared to the Knop model. As a result the heat fluxes computed using the Wilcox roughness model are the highest.

**Figure 15** Computed wall shear stress along the wall, condition 1 (see online version for colours)

![Wall shear stress](image1)

**Figure 16** Temperature (left) and Mach number (right) in the symmetry plane (see online version for colours)

![Temperature and Mach number](image2)
3.6.2 Results condition 2

Figure 18 shows the computed heat flux along the wall for condition 2 and the results are compared with empirical correlations for a laminar and turbulent flow provided by Airbus DS.

In the nose region \((y < 0.05)\), the calculated results show oscillations in the calculated heat flux that might be caused by the numerical scheme and the alignment of the shock with the grid lines. As for the calculations for condition 1, the calculated results show a transitional flow in this region when using the \(k-\omega\) Menter Shear Stress model, and this explains the higher heat fluxes obtained with this model. As to be expected, the largest roughness height yields the highest heat fluxes. The heat fluxes computed using the Spalart-Allmaras turbulence model show a laminar flow in the nose region, followed by transition to turbulence around \(y = 0.25\) m for the lowest roughness height and \(y = 0.2\) m for the largest one. Again, the highest heat fluxes are obtained for the largest roughness height.

For a laminar flow without blowing the empirical correlations yield heat fluxes that are about 10% higher than the corresponding CFD results. For the cases with roughness and blowing, it can be observed that using blowing model 2 yields the highest heat fluxes. Comparing the turbulent CFD results with the empirical correlations is difficult. The empirical correlations assume a laminar flow in the nose region, while the results obtained using the \(k-\omega\) Menter Shear Stress predicts a transitional flow. For this reason, heat fluxes obtained for this model are higher. Between \(y = 0.1\) m and \(y = 0.3\) m the empirical correlations predict the highest heat fluxes, while downstream of \(y = 0.3\) m the CFD results predict the highest heat fluxes, and large differences exists between the heat fluxes computed using the \(k-\omega\) Menter Shear Stress and Spalart-Allmaras turbulence models.
Figure 18  Computed heat flux along the wall, condition 2 (see online version for colours)

Figure 19 shows the computed wall shear stress along the wall for the different calculations. In the nose region until $y = 0.2$ m the computed wall shear stresses using the Spalart-Allmaras turbulence model are in good agreement with the empirical correlation. The wall shear stress computed using the $k-\omega$ Menter Shear Stress model is much higher.

Figure 19  Computed wall shear stress along the wall, condition 2 (see online version for colours)

Figure 20 shows the temperature contours in the symmetry plane. Maximum temperatures are around 8700 K just downstream of the shock.
3.6.3 Results condition 3

Figure 21 shows the computed heat flux along the wall in the symmetry plane for the calculations made for condition 3. An additional calculation was made using the Spalart-Allmaras turbulence model for a smooth wall without blowing to provide a reference. The heat fluxes from empirical correlations were provided by Airbus DS and are also shown in the figure. The computed heat flux obtained using the Spalart-Allmaras turbulence shows a laminar flow until around $y = 0.2$ m. The results obtained using the $k-\omega$ Menter Shear Stress turbulence model show a turbulent flow starting from the nose, and therefore the computed heat flux is substantially higher than the heat flux computed using the Spalart-Allmaras turbulence model. The empirical correlations predict a higher heat flux in the nose region than all CFD results.

Figure 22 shows the computed wall shear stress, together with the wall shear stress from the empirical correlation for a smooth wall. In the nose region, the computed wall shear stress using the $k-\omega$ Menter Shear Stress turbulence model is much higher than the results obtained using the Spalart-Allmaras turbulence model. As mentioned before, the calculations using the $k-\omega$ Menter Shear Stress turbulence model predict a turbulent flow.
starting at the nose, while for the calculations using the Spalart-Allmaras turbulence model transition to turbulence takes place around $y = 0.2$ m. The wall shear stress from empirical correlations is close to the computed results for the Spalart-Allmaras turbulence model until $y = 0.1$ m.

**Figure 21** Computed heat flux along the wall, condition 3 (see online version for colours)

**Figure 22** Computed shear stress along the wall, condition 3 (see online version for colours)

Figure 23 shows the computed temperatures in the symmetry plane. Maximum temperatures are around 3300 K in the stagnation region.
4 Conclusions

Calculations were made for the ERC configuration for three conditions on the re-entry trajectory. For each condition calculations were made using two turbulence models and for two different roughness heights. Computed heat fluxes and wall shear stresses were compared with results from empirical correlations provided by Airbus DS.

The computed results showed that for all three conditions the $k$-$\omega$ Menter Shear Stress model predicts a weakly turbulent flow at the nose, while the Spalart-Allmaras model (and the $k$-$\omega$ Wilcox model for condition 1) predicts a laminar flow in this region. As a result, the highest heat fluxes and wall shear stresses in the nose region are found using the $k$-$\omega$ Menter Shear Stress model.

For condition 1, for a laminar flow without blowing, the heat fluxes from CFD are in good agreement with the heat fluxes obtained from empirical correlations. For the same case with blowing the empirical correlation predicts slightly higher heat fluxes. A similar
observation can be made for the laminar flow results with blowing for condition 2. For condition 3, the heat fluxes obtained from the different empirical correlations are in the nose region substantially higher than the laminar and turbulent heat fluxes from CFD.

Comparing the computed heat fluxes using the Spalart-Allmaras turbulence model for the two roughness heights shows a much larger increase for the largest roughness height compared to the results from the empirical correlations and the results obtained using the $k-\omega$ Menter Shear Stress turbulence model.

The computed results showed that the heat flux and wall shear stress are increased with increasing roughness height. The influence of blowing is to reduce both the heat flux and wall shear stress. This reduction depends on the turbulence model employed, and is higher for the $k-\omega$ models. Blowing also leads to a faster transition to turbulence for the calculations that showed a laminar or weakly turbulent flow behaviour at the nose.

Comparing the roughness modelling for the $k-\omega$ Wilcox turbulence model showed that the Wilcox model leads to an earlier transition to turbulence and yields higher values of the heat flux and wall shear stress compared to the Knopp roughness model.

Computed heat fluxes along the wall strongly depend on the turbulence level in the boundary layer. The higher the turbulent viscosity in this region the higher the heat fluxes. It is therefore primordial to either know the transition location or that a turbulence model predicts the transition to turbulence.

Comparing the heat fluxes obtained using CFD for the cases with roughness and blowing to the heat fluxes from empirical show that in the nose region downstream of the transition point until the sphere-cone junction the empirical correlations predicts the highest heat fluxes. Downstream of the sphere-cone junction, the heat fluxes computed using CFD show an increase in heat flux, while the empirical correlations show a slight decrease. It should be remarked that in particular in the region from sphere-cone junction until the tore the heat fluxes from empirical correlations are not necessarily satisfactory because the entropy swallowing effect is not taken into account.

Two models to account for blowing are implemented in the empirical correlations, with model 2 yielding the highest heat fluxes.

As mentioned in the introduction, one of the objectives of the RASTAS SPEAR project was to increase knowledge that might lead to a reduction of uncertainties and design margins for sample return vehicles. The results of the CFD calculations presented in this paper raise several questions that require further study and validation through dedicated experiments before CFD can be used with confidence for the prediction of the heat fluxes of re-entry vehicles with an ablative TPS. For this reason, it is concluded that empirical correlations with sufficient design margins are still the recommended method for TPS sizing.

**Acknowledgements**

The work presented in this paper was carried out in the frame of the RASTAS SPEAR EU funded FP7 project, Grant Agreement No. 241992.

B. Aupoix is acknowledged for his valuable help with the implementation of the roughness modelling in the NSMB CFD code.
References


