25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

EVALUATION OF THERMAL EFFECTS ON THE SURFACE OF SARA SUBORBITAL PLATFORM DURING ATMOSPHERIC REENTRY

Tiago Gonçalves Faria
Carlos Eduardo Grossi Campos
Instituto Tecnológico da Aeronáutica (ITA) – Praça Mal. Eduardo Gomes, 50 - São José dos Campos -SP – Brazil
Instituto de Aeronáutica e Espaço - IAE, Praça Mal. Eduardo Gomes 50, São José dos Campos, SP, 12228-904, Brazil
tiago_gfaria@hotmail.com , carloseduardocegc@fab.mil.br

Humberto Araújo Machado
Instituto de Aeronáutica e Espaço - IAE, Praça Mal. Eduardo Gomes 50, São José dos Campos, SP, 12228-904, Brazil
Faculdade de Tecnologia – FAT/UERJ, Rod. Pres. Dutra km 298, Resende, RJ, 27537-000, Brazil
humbertoham@fab.mil.br

Abstract. *The prediction of the convective heat flux incident on blunt bodies reentry, as the SARA platform is of major importance to design the thermal protection of the spacecraft, which is responsible to ensure the integrity of the vehicle and payload during reentry flight. The engineering approach developed by Zoby was chosen to calculate the heat flux over the external surface of SARA. The results obtained by Zoby's Method were compared with the ones simulated using Computational Fluid Dynamics (CFD), which is a more complex method to estimate heating flux over the body surface and flow properties. Results show that Zoby's Method agrees with the numerical simulations when comparing the convective heat transfer coefficient and other flow properties at some specific trajectory instants. The study showed that Zoby's Method is a satisfactory methodology to assist in the design of the thermal protection of SARA platform respecting its limitations, reaching similar results achieved by numerical simulations performed by commercial CFD software.*

Keywords: *Zoby's method, blunt body, heat flux, atmospheric reentry, CFD.*

1. INTRODUCTION

Space and sub-orbital vehicles reach high velocities within atmosphere, and during reentry flight, major energy obtained is converted in high temperature, surpassing 2000ºC at the stagnation point. Ablative materials have been effectively used as Thermal Protection System (TPS) for spacecraft (Rogan and Hurwicz, 1973). The SARA (acronym for *SAtélite de Reentrada Atmosférica* – Atmospheric Reentry Satellite) is designed to offer good thermal behavior in both, surface and on-board devices, in order to preserve the payload requirements. SARA reaches the speed of 9300 km/h in atmospheric flight (Moraes, 1998). As a consequence, the SARA TPS is a critical aspect of the spacecraft design. This work focuses on the validation of an engineering method (Zoby et al., 1981) to predict the wall heat fluxes for laminar flow configurations when compared to commercial CFD software, ANSYS Fluent. The main objective is to study SARA surface reaction to dynamics reentry and its heating properties, with special care to the transition flow regime in the surface.

The Institute of Aeronautics and Space (Instituto de Aeronáutica e Espaço – IAE) of Brazil has designed, built and launched hundreds of rockets along the past 40 years. The VS-40 sounding rocket is a two-stage guided solid propellant rocket, with SARA payload, and is proposed to be used by Brazilian Space Agency (AEB) and European Space Agency (ESA). Figure 1 shows a schematic representation of VS-40. It has a total length of 7 m and a diameter of 1 m.

Figure 1. VS-40 Rocket.

To predict the heat flux on SARA, it is necessary previously know the pressure, temperature and velocity fields around the rocket. That can be accomplished by numerically solving Navier-Stokes equations. However, such a procedure is expensive and time consuming. In the present work a simpler, but reliable, engineering approach is used, which allows to obtain the convective heat transfer coefficient and the representative environment temperature for heat exchange without needing of a numerical solution of the Navier-Stokes equations.

Such simplifying assumptions are made below:

- Zero angle of attack and the rotation around its longitudinal axis is neglected;
- Physical properties are considered constant with temperature;
- Atmospheric air is considered to behave as a calorically and thermally perfect gas (no chemical reactions).

According to Machado (2012), these assumptions provide a good approximation to the real situation for the case of a ballistic reentry and for VS-40/SARA. These results will be compared with the numerical solution provided by the commercial CFD software ANSYS Fluent®, which is well-known software used to simulate supersonic and hypersonic flows.

2. METHODOLOGY

2.1 Zoby's Method

The Zoby´s Method (Zoby et al., 1981) seeks to predict convection heating of aerospace vehicles during reentry at hypersonic velocities (*M*>3). It is developed by means of laminar, transient and turbulent heating rate equations, considering the variable-entropy effects. The method currently established in the aerospace environment has proven to be applied for design of most prediction convective heating-rate distributions for the reentry vehicles of the IAE. Firstly, in order to predict the heat transfer on SARA, it is necessary to know pressure, temperature and velocity fields of the environment around the vehicle. The free stream conditions ahead of the SARA are those given by v_∞, T_∞, p_∞ (Velocity, temperature and pressure). By knowing v_∞ and altitude, as function of time, together with an atmospheric model (NOAA, 1976), it is possible to evaluate the free stream properties, such as p_∞, T_∞, and c_∞, which represent free stream pressure, temperature and speed of sound, respectively.

For supersonic flow ($M_\infty>1$), a shock wave appears ahead of the SARA nose. By using the normal shock relationships, it is possible to calculate v_1, T_1 and p_1 after shock.

The heat flux over the external surface was calculated through the Zoby's method (Zoby et al., 1981), as follows:

$$q = h(T_{aw} - T_w) \tag{1}$$

Where h is the convective heat transfer coefficient, q is heat flux and T_w é wall temperature.

The adiabatic wall temperature, T_{aw}, also called recovery temperature, T_r, and is given by:

$$T_{aw} = T_e + F_R \frac{{v_e}^2}{2c_p} \tag{2}$$

Where c_p is the specific heat at constant pressure, T_e is the temperature, v_e is the velocity, the subscript "e" refers to conditions at the boundary layer edge, F_R is the recovery factor, equal to $\sqrt{Pr_w}$, for laminar flow and $3\sqrt{Pr_w}$ for turbulent flow and Pr_w is the Prandtl number evaluated at wall temperature, $Pr_w = 0.71$.

The convective heat transfer coefficient by Reynolds analogy:

$$h = 0.5\rho_e c_p v_e Pr_w^a C_F \tag{3}$$

Where a is equal to 0.6 for laminar flow and 0.4 for turbulent flow; ρ is specific mass.

Assuming compressibility effects, a modified friction factor is obtained:

$$C_F = K_1 (R_{e\theta})^{K_2} \left(\frac{\rho_e^*}{\rho_e}\right) \left(\frac{\mu_e^*}{\mu_e}\right)^{K_3} \tag{4}$$

Where μ is the viscosity, the superscript "*" refers to properties evaluated at Eckert's reference temperature (T_e*) and $R_{e\theta}$: Reynolds number, based on the boundary layer thickness (θ):

$$R_{e\theta} = \frac{\rho_e v_e \theta}{\mu_e} \tag{5}$$

The viscosity μ, is evaluated according to Sutherland's equation, as function of temperature (Anderson, 1989). In Eq. 4, K_1 = 0.44, K_2 = -1 and K_3 = 1, for laminar flow. For turbulent flow, $K_2 = K_3 = -m$, and:

$$K_1 = 2\left(\frac{1}{C_5}\right)^{\frac{2N}{N+1}}\left[\frac{N}{(N+1)(N+2)}\right]^m \tag{6a}$$

$$m = \frac{2}{N+1} \tag{6b}$$

$$C_5 = 2.2433 + 0.93N \tag{6c}$$

$$N = 12.76 - 6.5log_{10}(R_{e\theta}) + 1.21[log_{10}(R_{e\theta})]^2 \tag{6d}$$

For laminar flow, the boundary layer thickness is given by:

$$\theta_L = 0.664\ \frac{(\int_0^y \rho_e^*\mu_e^* v_e R^2\ dy)^{\frac{1}{2}}}{\rho_e v_e R} \tag{7}$$

Where y (tangential coordinate) measured along the body's surface, y=0 corresponds to the stagnation point and R is the geometric parameter.

In this study, the numerical integration of Eq. 7 was obtained according to the trapezoidal method. As $R \to 0$, Eq. 7 becomes undetermined. By taking the limit of Eq. 7 as $R \to 0$, the following expression is obtained:

$$\theta_L = \frac{0.332(\rho_e^*\mu_e^*)^{\frac{1}{2}}}{\sqrt{\frac{1}{R_N}\left[\frac{2(p_2-p_\infty)}{p_S}\right]^{\frac{1}{2}}}} \tag{8}$$

In this study, Eq. 8 is applied for $y < 0.1\ R_N$, where R_N is the curvature radius at the stagnation point.

The boundary layer thickness for turbulent flow is obtained by solving the following first-order differential equation:

$$\frac{D(\rho_e v_e R\theta_T)}{D_y} = 0.5C_F\rho_e v_e R \tag{9}$$

After obtaining the boundary layer momentum thickness, θ, Re_θ, C_F and h can be evaluated by using equations 3, 4 and 5. Along the transition region between laminar and turbulent flow, the following relationship is used (*Zoby et al., 1981*):

$$q_{Tr} = q_L + F(y)(q_T - q_L) \tag{10}$$

Where the subscripts Tr, L and T represent, respectively, transitional, laminar and turbulent flow. The transitional factor, $F(y)$, is given by:

$$F(y) = 1 - exp\{-0.412\left[\frac{4.74(y-y_L)}{(y_T-y_L)}\right]\} \tag{11}$$

Transition is supposed to occur for $163 < Re_\theta < 275$.

Property evaluation at the boundary layer edge is performed assuming isentropic flow between the stagnation region and the location i, where properties are needed:

$$\rho_{e,i} = \rho_s\left(\frac{p_{e,i}}{p_s}\right)^{\frac{1}{y}};h_{e,i} = h_s\left(\frac{p_{e,i}}{p_s}\right)^{\frac{y-1}{y}};v_{e,i} = \sqrt{2(h_s - h_{e,i})};T_{e,i} = \left(\frac{h_{e,i}}{C_P}\right) \tag{12}$$

Where i is discrete position in which the parameter is calculated along the surface. At the stagnation point, i = 1.

The local pressure, $p_{e,i}$, is obtained from the modified Newton method and γ = 1.4. The results of both methods are then compared. The subscripts appearing in Eq. 12 refers to the stagnation condition. Eckert reference temperature is obtained from:

$$\frac{T_{e,i}^{*}}{T_{e,i}} = 1 + 0.032M_{e,i}^{2} + 0.58\left(\frac{T_w}{T_{e,i}} - 1\right) \quad (13)$$

The solution procedure can be summarized as follows:

From a given trajectory, the US Standard Atmosphere (National Oceanic and Atmospheric Administration, 1976) is used to obtain the free stream properties, including the stagnation point.

- Normal shock relationships are used to obtain the fluid flow properties behind the shock.
- By using the modified Newton method, pressure distribution is obtained along the payload.
- Equation 12 provides the local properties at the boundary layer edge.
- If $y < 0.1\ R_N$, Eq. 8 provides the laminar boundary layer thickness, leading to the estimation of $R_{e\theta}$, Cf and h, provided by Eqs. 3, 4 and 5, respectively.
- If $y > 0.1\ R_N$ and $R_{e\theta} < 163$, Eq. 7 is numerically integrated up to the location where the momentum thickness is to be estimated. Such integration is performed by using the trapezoidal method.
- If $y > 0.1\ R_N$ and $R_{e\theta} > 275$, Eq. 9 is numerically integrated by the trapezoidal rule leading to the turbulent boundary layer thickness.
- If $y > 0.1\ R_N$ and $163 < R_{e\theta} < 275$, Eqs. 10 and 11 are used to estimate h.

It is important to emphasize that such a procedure is performed along the payload's surface following the *y*-coordinate for different trajectory times. The coordinate system used in this calculation is described in Fig. 2.

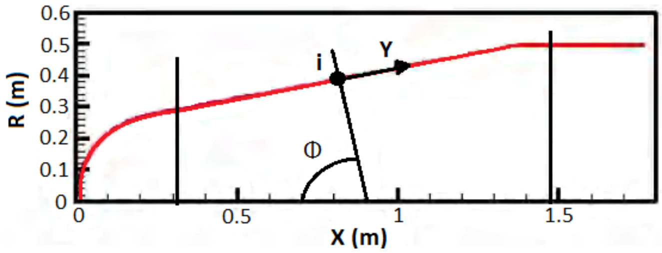

Figure 2. SARA platform coordinate system.

2.2 Numerical Simulation

To validate the results obtained by the Zoby Method, numerical simulations were conducted to solve the Navier-Stokes equations to calculate the incident heat flux for the laminar flow using the commercial software ANSYS Fluent®. The solver used the Reynolds-averaged Navier-Stokes equations (RANS) to simulate the laminar flow around the capsule surface. Due to its geometrical characteristics, an axisymmetric flow was considered.

The implemented boundary conditions were prescribed wall temperature of 300 K for the vehicle surface, axisymmetry around the stagnation point and far field for the other boundaries representing the atmospheric free flow conditions. The external flow properties were calculated based on the *1976 U.S. Standard Atmosphere* and the SARA trajectory data for some specific instants, showed on Tab 1.

Table 1. SARA trajectory data for two instants.

Time (s)	Altitude (km)	Mach
50	56.442	7.826
43	39.428	7.508

These two instants from SARA trajectory were chosen due to their flow characteristics, where at 50s the flow presents a laminar boundary layer behavior and at 43s the flow presents a turbulent boundary layer over his surface. Considering both instants, it is possible to compare the laminar, transitional and turbulent formulation of Zoby's method with CFD simulations.

Fig. 3 shows the boundary layer Reynolds number calculated through Zoby's Method. It is possible to observe that for instant 50 seconds the Reynolds number value are always bellow 163, which means it will have a laminar behavior along the surface. For instant 43s, the Reynolds number reaches values up to 606, which means it will have a laminar, transitional and turbulent behavior along the platform surface. For the numerical simulations, first it was considered a laminar formulation to represent the flow condition at 50s and for the instant 43s. A turbulence model k-epsilon (κ-ε) was chosen to represent the turbulent flow characteristics.

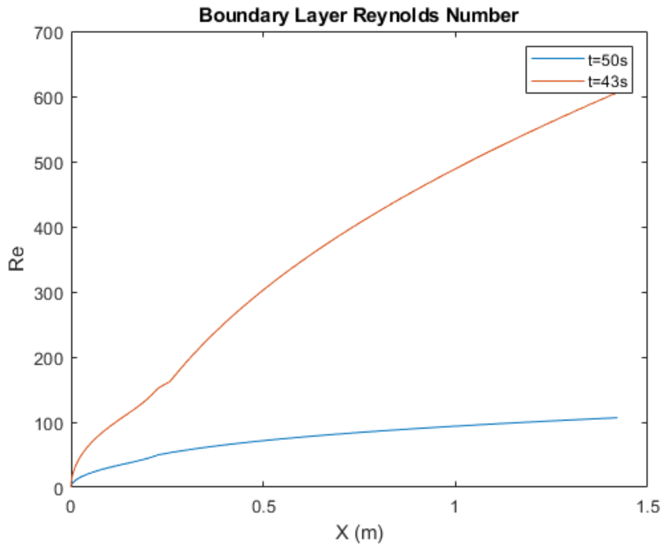

Figure 3. Boundary layer Reynolds number over SARA surface (Zoby's Method).

The computational mesh developed for the simulations has around 22.000 structured elements and for each different Mach number that was considered, a different point distribution was set to align with the shock wave direction and increase the numerical stability. Figure 4 shows one of the meshes used in the simulation.

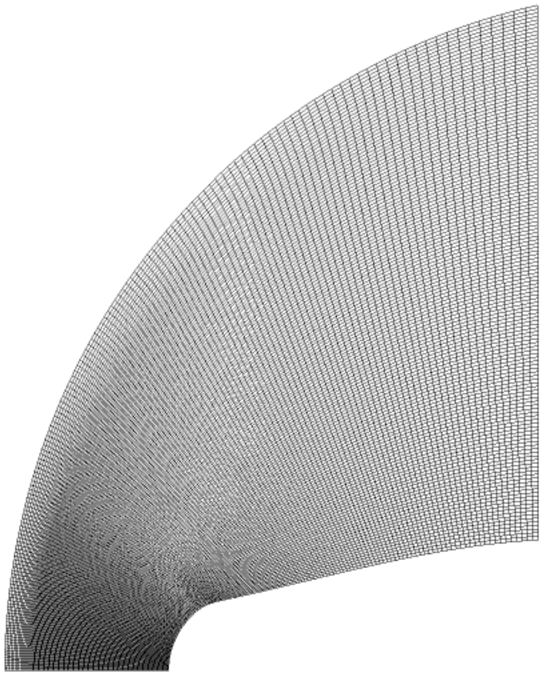
Figure 4. Computational mesh with 22.000 elements.

3. RESULTS

The pressure distribution along the vehicle surface was calculated for Mach 7.8 and 56.442 km altitude according with the engineering approach, for instant 50s (see Tab. 1). The achieved results were compared with the numerical simulations and the good agreement between them is presented in Fig. 5. The results were dimensionless related to the pressure value at the stagnation point. All the following graphs consider the X- coordinate as the axisymmetric coordinate o the platform.

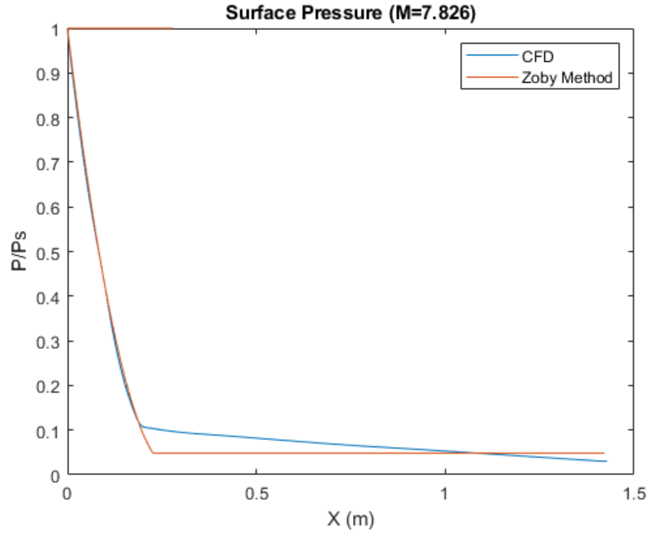

Figure 5. Pressure distribution along SARA surface for Mach 7.826 and 56.442 km altitude.

From the numerical analysis is possible to plot the velocity and the temperature contour along the flow field in the vicinity of the platform. These contours are shown on Fig. 6 for Mach 7.826 and 56.442 km (see Table 1).

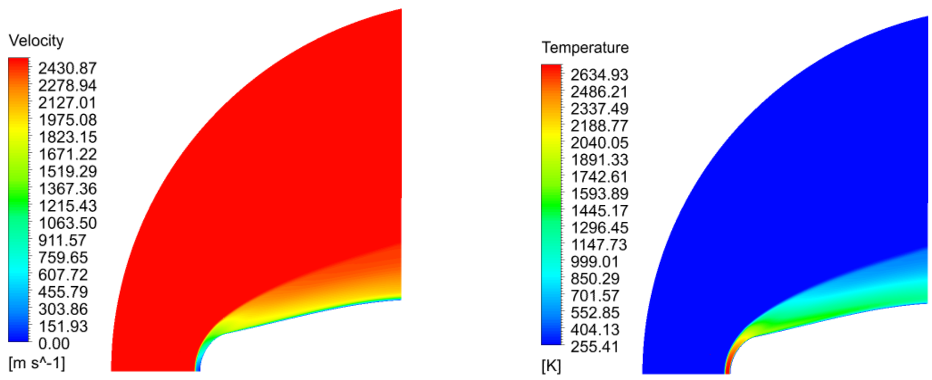

Figure 6. Velocity and temperature contour for Mach 7.826 and altitude 56.442Km.

The heat flux distribution along SARA surface is presented in Fig. 7. for Mach 7.826, where Zoby's method and CFD results show similar results. The heat flux values were dimensionless related to the heat flux value at the stagnation point. The laminar boundary layer flow characteristic implemented by the analytical methodology was crosschecked with the numerical results for laminar formulation and both agreed with similar results for heat flux distribution along the surface. It is important to notice that for lower Mach number values, the analytical methodology is not trustworthy and some nonphysical behaviors can appear. This expected behavior is well explained by Zoby (Zoby et al., 1981).

For very high Mach number, the assumption of non-chemical reactions is not valid anymore and results are influenced, needing some extra formulations to include this phenomenon and achieve more realistic results. Also, for the calculations, air was considered as a perfect gas. Nevertheless, this approach is more conservative and overestimates the results when compared to considering air in chemical equilibrium, which is a more realistic formulation.

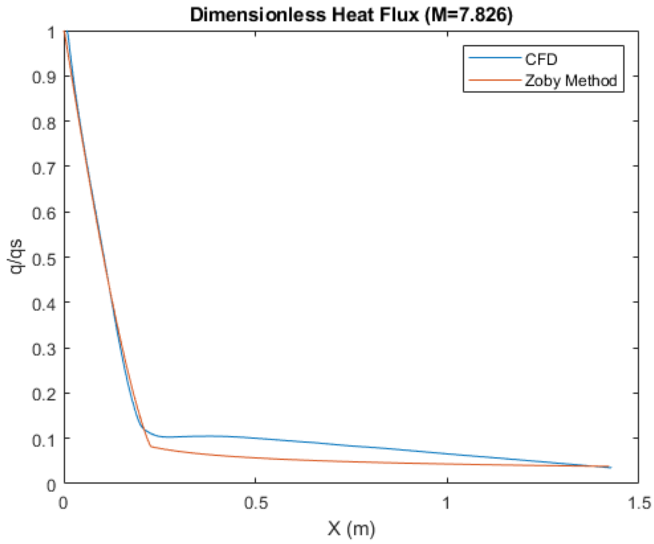

Figure 7. Heat flux on SARA surface calculated by Zoby and CFD for Mach 7.826 and 56.442 km altitude.

For the SARA trajectory instant 43s, the flow has a turbulent behavior as previously mentioned. Junqueira (Junqueira Jr. C., 2013) simulated the heat flux over SARA platform considering only as a laminar flow for all instants, in that case, the turbulent formulation of Zoby's method was not validated for the altitudes where the Reynolds number were high and the flow was turbulent.

Numerical simulations were performed to compare the results for heat flux distribution over the surface with Zoby's method, in this case, also considering turbulent formulation for both calculations, engineering method and numerical simulation. Fig 8 compares the dimensionless heat flux results for CFD and Zoby's method simulations. It is possible to notice a good agreement between the curves. It is also noticed the presence of a step along the Zoby's method curve. This step represents the instant where the boundary layer flow changes from laminar to turbulent formulation, the implemented transitional formulation links both expressions but its representation is not as smooth as the numerical simulations. The simulations performed by ANSYS Fluent® including the k-epsilon turbulence model has a more accurate and smooth behavior when compared to the engineering approach, due to its differences, there will be a small discrepancy between results on the turbulent portion of the flow. Even though the small differences between the results, it is possible to state that Zoby's method is also satisfactory to predict heat flux for flows with turbulent boundary layer.

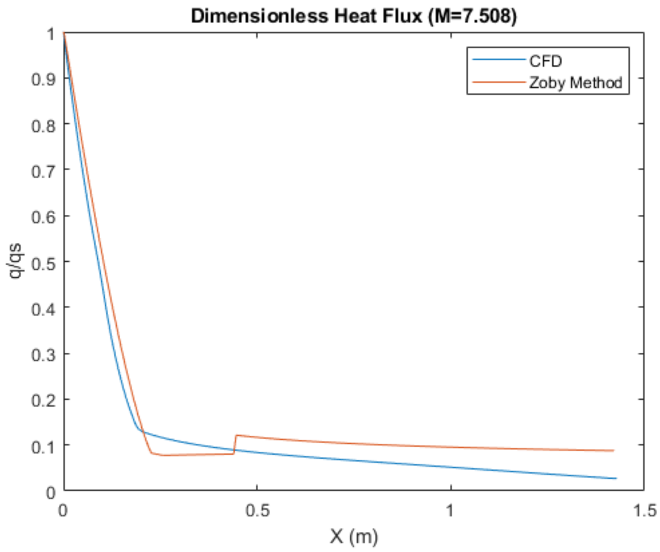

Figure 8. Heat flux on SARA surface calculated by Zoby and CFD for Mach 7.508 and 39.428 km altitude.

4. CONCLUSION

The engineering approach proposed by Zoby was compared with the numerical simulation results, focusing in the specific limitations that the analytical method must respect. For low Mach numbers values, CFD methodology is capable of performing correct predictions of the heat flux over the body surface while the engineering approach is not applicable for this Mach range, which is proofed by Zoby (Zoby et al., 1981).

Also, it is important to notice that for high Reynolds number values (Re $>10^6$), the flow will present turbulent effects and the laminar assumption is no longer valid, which will affect the heat flux distribution along the wall surface. For this reason, turbulence modeling must be included in numerical simulations and also in Zoby's method. CFD simulations with k-epsilon turbulence model were compared to the turbulent boundary layer methodology implemented by Zoby for the engineering approach. Both results presented a satisfactory agreement once comparing heat flux distribution. Another matter to notice is for the conditions with high Mach number, where the chemical reactions will appear due to its high temperatures and a formulation to calculate the influence of ionic dissociation must be included. Also, for more realistic results, air should not be considered as an ideal gas, but as a gas in chemical equilibrium.

Zoby's method is a fast engineering method that calculates heat flux and other thermal properties along surface of hypersonic blunt bodies This method presents some geometrical and flow velocity constraints, but for its range of applicability, it was substantiated that this methodology can be applied for a reentry capsule as SARA.

5. REFERENCES

Anderson Jr., J.D., 1989, "Hypersonic and High Temperature Gas Dynamics", McGraw-Hill, New York, USA.

Junqueira Jr. C., 2013, "Calculation of Wall Heat Fluxes for the Sub-Orbital SARA Capsule Atmospheric Flight", *22nd International Congress of Mechanical Engineering*, RibeirãoPreto, Brazil.

Machado, H.A., 2012, "Two-Dimensional Simulation of Ablation due to Aerodynamic Heating in the SARA Sub-Obital Platform". *High Temperatures -High Pressures.* Vol.41, pp. 255-271.

Miranda, I. F. e Mayall, M. C. M., 2001. Fluxo de Calor Convectivo em Micro-Satélites em Reentrada Atmosférica. Undergraduation thesis, ITA, São José dos Campos, Brasil.

Moraes Jr., P., 1998, "Design Aspects of the Recoverable Orbital Platform SARA", *Proceedings of 8th Chilean Congress of Mechanical Engineering*, Concepcíon, Chile.

National Oceanic and Atmospheric Administration – NOOA, 1976, "U.S. Standard Atmosphere, 1976", U.S. Government Printing Office, Washington, USA.

Oliveira, U. C., 1996. *Fluxo Convectivo Aerotermodinâmico em Corpos de Revolução com Ângulo de Ataque em Escoamento Supersônico*. MSc dissertation, ITA, São José dos Campos, Brasil.

Rogan. J.E. and Hurwicz, H., 1973, "High-temperature Thermal Protection Systems," *Handbook of Heat Transfer*,

edited by Rohsenow, W.M. and Hartnett J.P., McGraw-Hill, New York.

Zoby, E. V., Moss, J. N. and Sutton, K., 1981. "Approximate Convective Heating Equations for Hypersonic Flows". *Latin American Journal of Solids and Structures*, Vol. 18, No. 1, pp. 64-70.

6. RESPONSIBILITY NOTICE