23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

THERMAL PERFORMANCE EVALUATION OF TISSC THERMAL PROTECTION SYSTEM IN THE SARA SUB-ORBITAL

Humberto Araujo Machado
Instituto de Aeronáutica e Espaço – IAE, Pç Mal Eduardo Gomes, 50, São José dos Campos, SP, 12228-904
Faculdade de Tecnologia - FAT, Universidade do Estado do Rio de Janeiro - UERJ, Rodovia Presidente Dutra, km 298 (sentido RJ/SP), Pólo Industrial, Resende, RJ, 27537-000
humbertoham@iae.cta.br

Carlos Eduardo de Souza
Univesidade Federal de Santa Maria
e-mail

Ricardo Fortes de Miranda
Federal University of Uberlândia, School of Mechanical Engineering, Av. Joâo Naves de Ávila, 2160, Campus Santa Mônica, Bloco 1M, Uberlândia, MG, 38400-902
rf.miranda@mecanica.ufu.br

Abstract. *In this work, the use of a Thermally Integrated Structural Sandwich Core (TISSC), a new kind of space vehicle thermal protection is proposed, replacing the conventional cork layer used as a TPS (Thermal Protection System). The TISSC is applied to the SARA sub-orbital platform, which is being developed by IAE/CTA for micro-gravity research. The advantages of using such a structure are its lightweight and the same insulation as load bearing capabilities, low maintenance, and low life-cycle cost. A specific layout for this system is studied via computational simulation, and the thermal performance with diverse materials is evaluated. Results show the presence of hot spots in the internal wall, where spars linking the external and internal walls are placed.*

Keywords: *Thermal protection system, Aerodynamic heating, Structurally integrated thermal protection, Numerical simulation*

1. INTRODUCTION

Space and sub-orbital vehicles reach high velocities within the atmosphere, which results in aerodynamic heating and wall high heat fluxes. The high temperatures may negatively affect the structures mechanical behavior, the operation of on board equipment, besides the fact that payload preservation is always mandatory. These problems are solved by using an efficient Thermal Protection System (TPS). As a consequence, TPS design is a critical aspect of a rocket design, since under sizing may result loss of the payload, and the over sizing it implies in increasing mass and cost. Sub-orbital platforms are a low-cost alternative for micro-gravity research. The SARA sub-orbital platform is being developed by IAE/CTA for such application. It has a total mass of 250 kg for a payload of about 25 kg, and is is designed to provide 6 minutes of micro-gravity environment. SARA reaches the speed of 9300 km/h in atmospheric flight (Moraes Jr., 1998). The current design uses a conventional cork layer as TPS to protect its lateral surface, with the trade-off of large mass. As an option, Thermally Integrated Structural Sandwich Core (TISSC) would insulate the vehicle from aerodynamic heating as well as carry primary vehicle loads (Martinez et al., 2005). It consists of a structural sandwich panel in a three-layer plate with two face sheets and a core. Two thin, stiff and strong faces are separated by a thick, light and weaker core. The advantages of using such a structure are that it is lightweight, multifunctional, for example offer insulation as well as load bearing capabilities, low maintenance, and low life-cycle cost. Commonly used materials for facings and core are ceramic matrix composite laminates and metals. In this work a Thermally Integrated Structural Sandwich Core (TISSC) is proposed to replace SARA's current thermal protection system. Thermo-structural analyses of the proposed configuration are performed, and the results are compared against those obtained for the current TPS design.

A Finite Element Method (FEM) commercial software, ABAQUS, is employed to perform the analyses. This software is able to perform only thermal or coupled thermo-structural analyses. It allows the application of user subroutines for various purposes, such as specifying the heat flux on a surface as function of time. Next, the physical problem and the mathematical model are discussed, followed by the description of the numerical studies performed.

2. PHYSICAL PROBLEM AND MATHEMATICAL MODEL

To predict the heat transfer on SARA, it is necessary to know pressure, temperature and velocity fields around the vehicle. That can be accomplished by numerically solving the N-S equations. However, such a procedure is expensive and time consuming. In the present work a simpler, but reliable, engineering approach is also used.

The heat flux over the external surface was calculated through the Zoby's method (Zoby et al., 1981). Once the convection heat transfer coefficient and the adiabatic wall temperature are known, wall temperature distributions can be obtained. For the previous arrangement of thermal protection, is a simple heat conduction problem. Since the wall is made with Aluminum, which has a high thermal diffusivity, it can be treated as a one-dimensional problem along the wall thickness (Machado, 2006).

The core is filled with Saffil, an alumina fibrous insulation. The thermal conductivity of Saffil was obtained from the model of Daryabeigi (2003) for heat conduction in porous media. The mathematical model used is detailed described in Daryabeigi's work. The model considers the heat transfer between two parallel plates, combining the radiation and conduction modes. The fibrous insulation between the plates is considered a participating media. The main equation resulting from the assumptions is the one-dimensional energy conservation equation, with a source term for the radiation heat transfer.

3. NUMERICAL STUDIES

In this work, two basic models are investigated. The first one is a flat brick model, representing the sandwich proposed by Machado et al. (2013). The applied thermal loading was presented by Junqueira-Junior et al. (2013), and are plotted in Fig. 1. This flat brick is composed by two plates separated by an 18 mm thick layer of Saffil ®. That study considered a heat flux on the external wall defined by the equation:

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

where q is the heat flux, h is the heat transfer coefficient, T_w is the wall temperature and T_r is the recovery temperature (also called as adiabatic wall temperature).

As seen in eq. 1, the heat flux depends also on the wall temperature. In the finite element code, an iterative process is executed, where the heat flux on step k depends on the T_w obtained on step $k - 1$. In Abaqus, the user routine DFLUX is employed for that end. The reader is referred to the Abaqus online manual for more information Simulia (2013).

Table 1 shows the mechanical and thermal properties of materials considered in the present study.

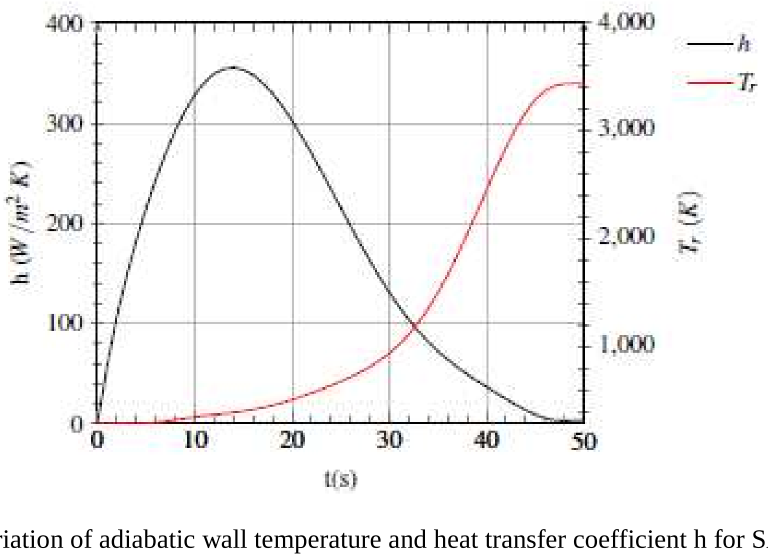


Figure 1. Time variation of adiabatic wall temperature and heat transfer coefficient h for SARA's trajectory.

Table 1. Material properties used in the Finite Element model.

Property	Aluminum	Stainless steel	Carbon fiber	Saffil	
density ρ	2710	8000	1600	24.2	kg/m^3
specific heat C_p	960	500	800	1005	(J/kg K)
thermal conductivity k	177	16.2	150	var.	(W/mK)

3.1 Flat-plate sandwich

A first analysis is used to validate the numerical simulation. A square flat-plate sandwich is modeled with Abaqus, trying to reproduce the analytical results obtained analytically by Machado et al. (2013). Figure 2 shows the simulated geometry and cross section. An initial temperature of 27^o C is applied to the whole model, and a time dependent heat flux is applied to the upper wall surface. The heat flux is computed by the DFLUX user routine in Abaqus, using Eq. 1.

A very good agreement with the previous values is obtained, as seen in Fig. 3. Despite a small transient difference for the internal temperature, the final values are within 0.5^{o}C distance to the analytical values.

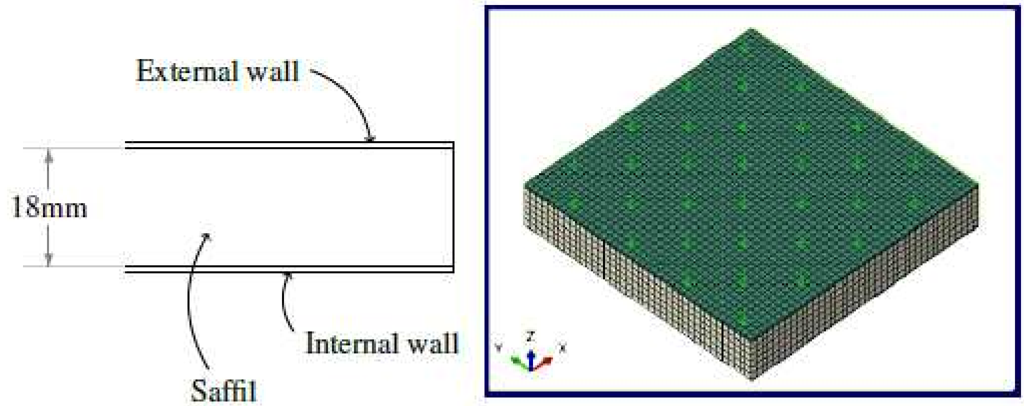


Figure 2. Flat-plate sandwich model. On the right, the applied heat-flux on the external wall.

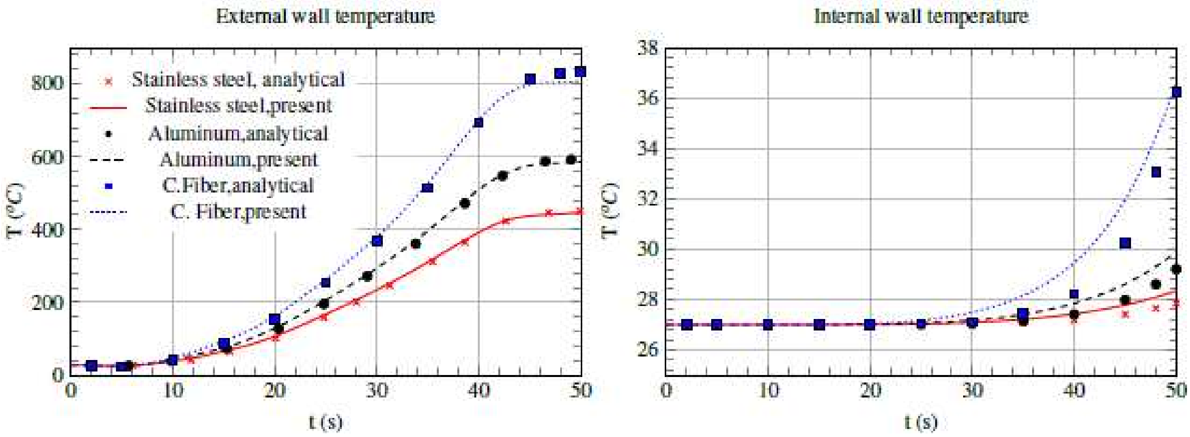


Figure 3. Comparison between analytical and present results for external (left) and internal (right) wall temperature.

3.2 Conical 3-D Model

The basic 3-D model consists of a quarter part of the conical walls of SARA's payload bay section, shown in Fig. 4. The internal wall is modeled with aluminum and the external wall with stainless steel.

A variable number of 1mm thick longitudinal spars connect the external and internal walls. The sandwich core consists of Safill, filling the empty space between all structural parts. With this configuration, it is expected that the temperature on the internal wall will present a peak at the spars position.

The FEM model is assembled so that the material of these spars can be modified independently of the internal and external walls. This way, it is possible to evaluate the peaks for different materials. A reference condition for heat transfer analysis only is simulated using the properties of Safill, what represents a simple sandwich with no structural reinforcement. This condition is not structurally efficient, serving only for comparison of temperature fields on the walls.

The table in Fig. 4 shows 3 geometric variations of the basic model, with different values of external wall thickness and number of spars. The boundary conditions are the same used in the flat-plate sandwich model.

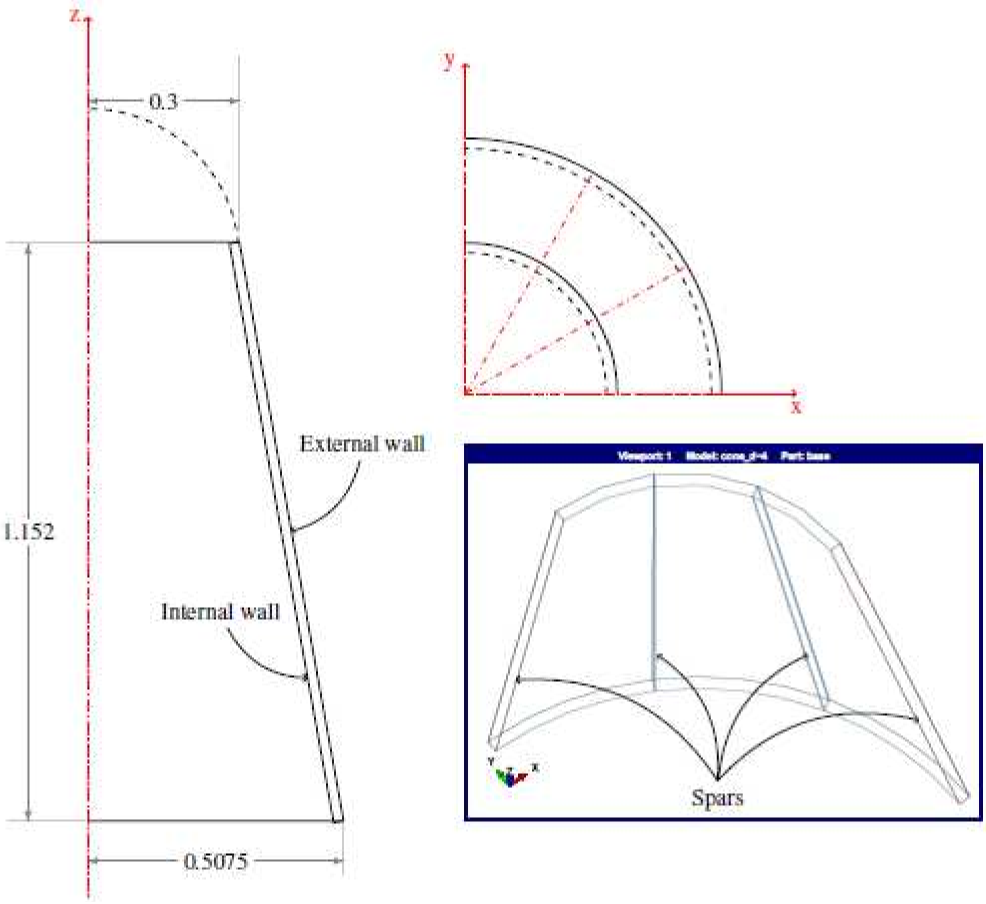


Figure 4. Dimension of SARA's conic section and position of spars.

Table 2. Configurations tested. For all cases, the internal wall material is aluminum and the external wall is stainless steel.

Config.	internal wall thickness	external wall thickness	number of spars	Total mass: Only Saffil (kg)	Total mass: S.Steel spar (kg)	Total mass: Aluminum spar (kg)	Total mass: Carbon F. spar (kg)
A	1.0 mm	1.0 mm	4	8.027	9.048	8.371	8.229
B	1.0 mm	0.5 mm	3	5.115	5.795	5.344	5.250
C	1.0 mm	0.5 mm	5	5.115	6.475	5.573	5.384
D	0.5 mm	0.5 mm	4	4.174	5.193	4.517	4.376

Figure 5 shows the temperature field for three reference condition of configuration A (1 mm internal and external walls) with only Saffil and no reinforcement spars. As expected, the temperature is homogeneous on the walls. On the other side, Fig. 6 shows the results for the models with the presence of spars made of stainless steel, aluminum and carbon fiber. It is clearly observed the areas around the reinforcements present higher gradients along the walls.

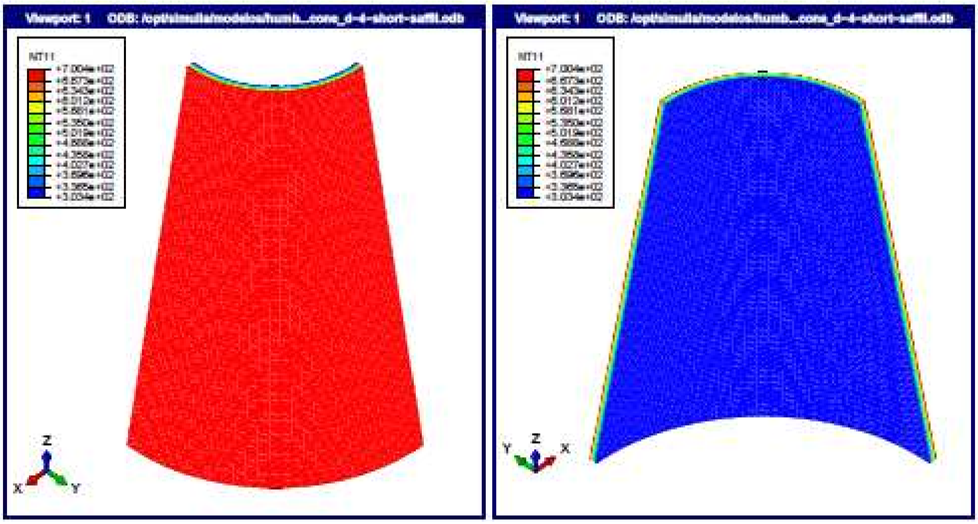
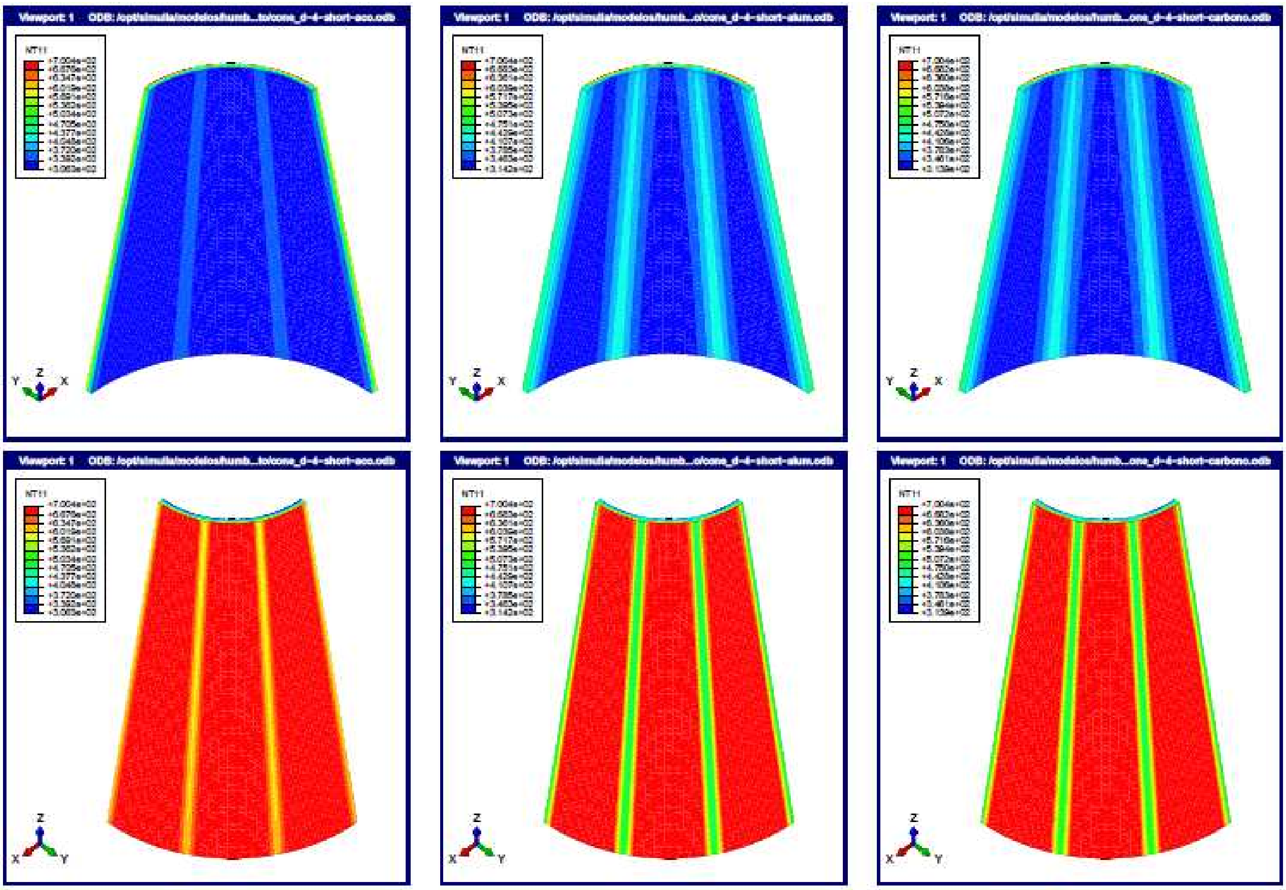


Figure 5. Reference temperature field on internal and external walls - Config. A, only Saffil.



(a)Steel (b)Aluminum (c)Carbon F.

Figure 6. Temperature field on internal and external wall - Config. A.

The graphics on Figures 7, 9, 8 and 10 show the temperature along the upper and bottom edges of internal and external walls for configurations A, B, C and D. For each configuration the curves for different spar material are compared to the condition with Saffil only. The peaks at the spars locations are very strong. For the cases with aluminum or carbon fiber spars, the external and internal walls temperatures coincide. This effect shows that the insulation is not adequate if the spars are made of aluminum or carbon fiber.

Configuration A has the lower external wall temperature, what is expected because of its larger thickness, that is 1mm against 0.5mm of other configurations. Another condition observed is that with more spars, the internal wall average temperature increases, reaching values near 100° C. A test that must be performed consists in modifying the spars thickness. Here, all are 2 mm thick.

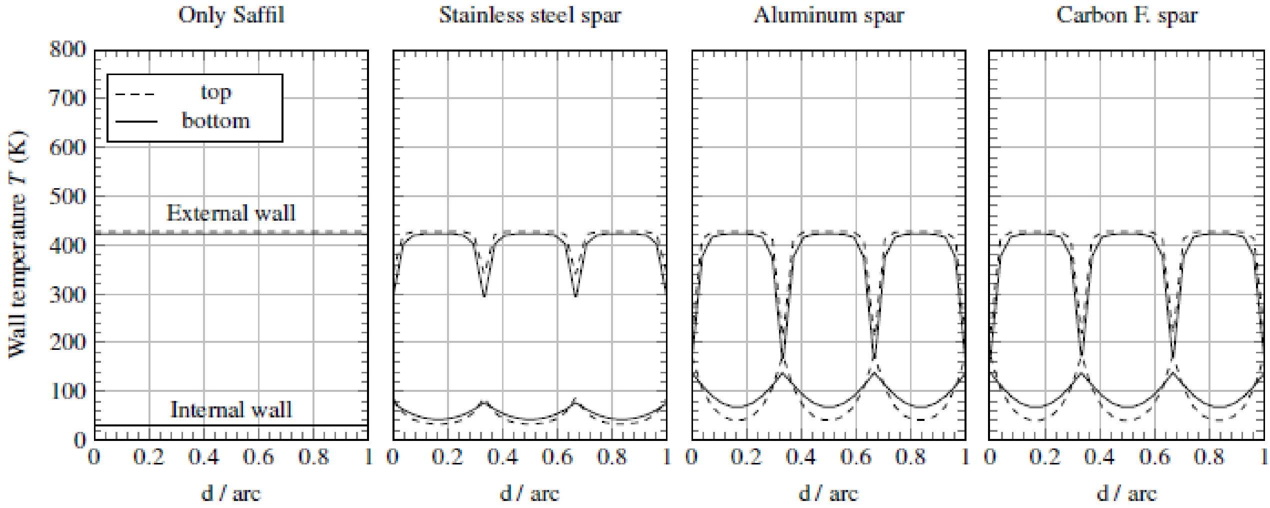


Figure 7. Temperature along upper and bottom edges of external and internal walls - Config. A.

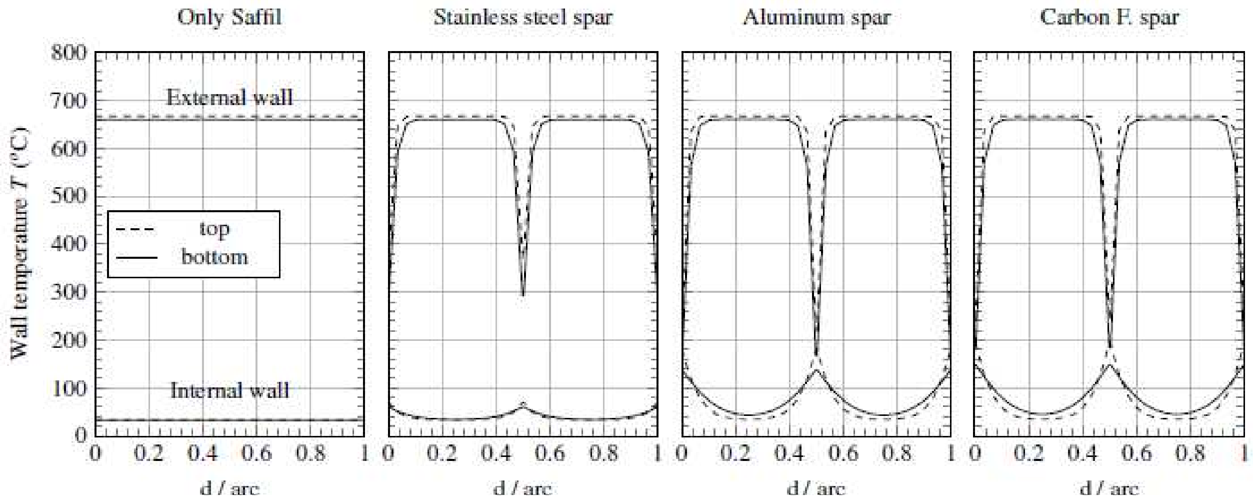


Figure 8. Temperature along upper and bottom edges of external and internal walls - Config. B.

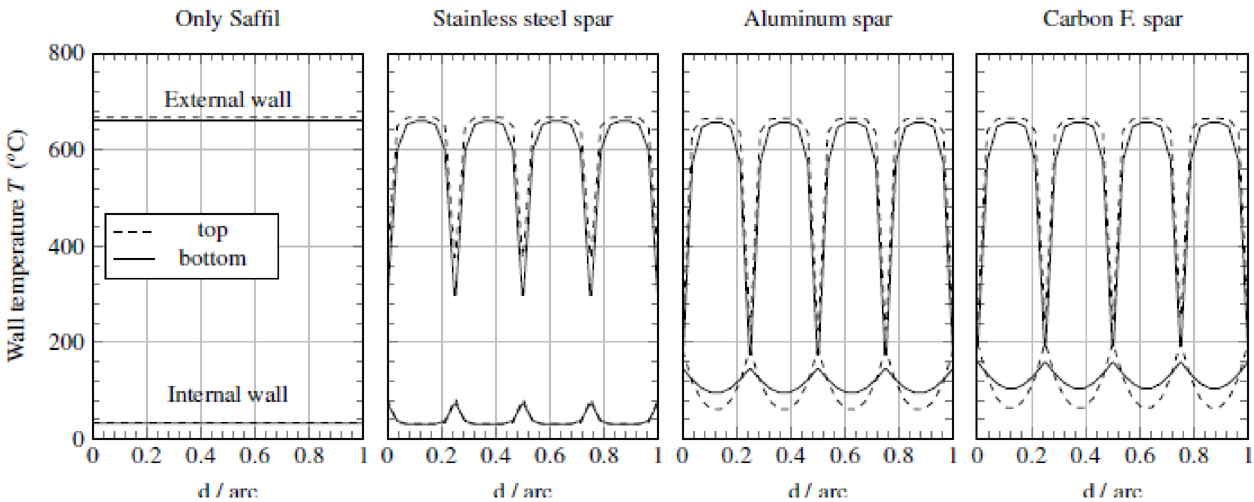


Figure 9. Temperature along upper and bottom edges of external and internal walls - Config. C .

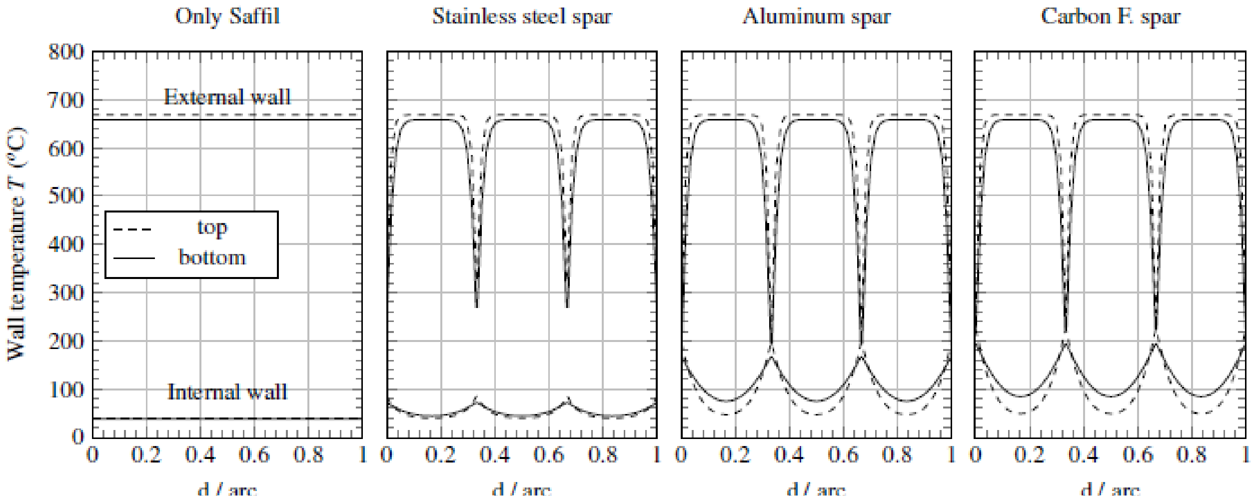


Figure 10. Temperature along upper and bottom edges of external and internal walls - Config. D.

4. CONCLUDING REMARKS

In this work, the thermal performance of a TISSC protection system was applied to SARA suborbital platform, and the temperature profile were obtained via computational simulation, employing the FEM-base software Abaqus ©. After validation, results were obtained for a few combinations of materials in the proposed configuration. It is observed the need of thermal insulation capabilitty between the external and internal surfaces, provided by the spars, in order to reduce the temperature peaks where these elements are placed.

5. ACKNOWLEDGEMENTS

Authors would like to thank CNPq and AEB (Brazilian Space Agency) for the financial support during this work.

6. REFERENCES

Daryabeigi, K., 2003. “Heat Transfer in High Temperature Fibrous Insulation”. *Journal of Thermophysics and Heat Transfer*, Vol. 17, No. 1, pp. 823–832.

Junqueira-Junior, C., Barreto, F.R.C., Azevedo, J.L.F., Machado, H.A., Basso, E. and Damilano, J.G., 2013. “Calculation of wall heat fluxes for the sub-orbital sara capsule atmospheric flight”. In *Proceedings of the 22nd International Congress of Mechanical Enginering (COBEM 2013)*. Uberlândia, MG.

Machado, H.A., 2006. “Thermal Protection for Aerodynamic Heating in The SARA Sub-Orbital Platform”. In *Proceedings of the IV National Congress of Mechanical Engineering - CONEM*. Recife, Brazil. (in Portuguese).

Machado, H.A., Pires, M., Ameloti, G.B. and de Miranda, R.F., 2013. “Structurally Integrated Thermal Protection Applied in The SARA Sub-Orbital Platform”. In *Proceedings of the XXXIV CILAMCE*. Pirenópolis, Brazil.

Martinez, O.A., Bapanapalli, S., Sankar, B., Haftka, R. and Blosser, M., 2005. “Micromechanical analysis of composite truss-core sandwich panels for integral thermal protection systems”. In *47th AIAA/ASME/ASCE/AHS Structures, Structural Dynamics and Materials Conference*. Newport, RI.

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

Simulia, 2013. “Abaqus 6.13 Online Documentation”, Dassault Systémes, Providence, RI, USA. <http://50.16.176.52/v6.13/>.

Zoby, E., Moss, J. and Sutton, K., 1981. “Approximate Convective Heat Equations for Hypersonic Flows”. *Journal of Spacecraft and Rockets*, Vol. 18, No. 1, pp. 64–70.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.