# ENCIT-2018-0388
# ANALYSIS OF HEAT TRANSFER IN AN ACADEMIC ROCKET ENGINE USING NUMERICAL METHODS

**Felipe Pinheiro Maia**
**George Santos Marinho**
Federal University of Rio Grande do Norte. Department of Mechanical Engineering. Avenue Senador Salgado Filho, S/N. Mailbox 1524 - Campus Universitário - Lagoa Nova, CEP 59072-970, Natal-RN, Brazil.
felipepm99@hotmail.com, gmarinho@ct.ufrn.br

*Abstract. The aerospace sector has great importance for the technological development of a nation, since it has diverse applications, from the area of telecommunications to questions of the defense of the national sovereignty. Thus, the study of space vehicles, and their design complexities, is necessary. One of the main problems in the development of rockets is the temperature, which reaches very high values in the combustion chamber of the rocket engine, which ends up creating a high temperature gradient and, consequently, facilitating the heat transmission to parts of the fuselage, which may not tolerate marked heating due to their material. In this work, numerical methods was used to thermally analyze the heat transfer of a class J engine to the fuselage, built in composite, of a 1500 m apogee academic rocket. A maximum temperature of 80°C was established in the walls of the body tube, due to the Glass Transition Temperature, so that the objective was to find, through the simulation of several insulation configurations, the configuration that would best maintain the structural integrity of the rocket. Graphs of the temperature distribution were obtained along the analyzed model, as well as graphs of validation of the results. It was concluded that the best configuration was the use of timber as structural material and fiberglass as insulator.*

*Keywords: Numerical Methods, Thermal Analysis, Heat Transfer, Rockets, Thermal Insulation*

## 1. INTRODUCTION

The aerospace industry has unique relevance in the technological development of a nation. Its applications range from areas such as telecommunications, sensing and monitoring to issues of defense of national sovereignty. In this sense, it is necessary to constantly study the vehicles of access to space, the rockets, especially in relation to the various complexities found in their projects, in order to improve the technologies and, consequently, to make the missions to outside the atmosphere safer and with lower cost.

One of the recurring concerns of a rocket designer is the temperature. As pointed out by Miranda and Pessoal Filho (2001), the effect of aerodynamic heating, which occurs due to the conversion of the energy of flow into heat, is one of the main problems encountered in space vehicles. To remedy this problem, some types of thermal protection systems can be used, such as ablation, reradiation, transpiration and regeneration (Costa and Moraes Jr, 2001). Another thermal problem can be observed in rocket engines. The chamber temperature of a solid-fuel rocket engine can exceed 3000 K after ignition (Guo et al., 2006), which represents a high risk of damage to the engine and rocket structures, since that depending on the manufacturing materials of each part, its mechanical strengths can vary considerably. Thus, thermal analyzes are fundamental during the rocket design stage.

Since analytical methodologies for heat transfer analysis have their application limited by resolution capacity of system governing equations, it is common to use numerical methods for analysis of more complex configurations. One of them, widely used in the most diverse areas of engineering, is the Finite Element Method (FEM). According to Chandrupatla and Belegundu (2011), this method consists of discretizing a continuous complex region into small elements of simple geometry. On these elements are applied the properties of the materials, the system governing equations and the boundary conditions, in order to obtain a system of equations that can be solved and whose solution is the behavior of the region in relation to some parameter, such as temperature. Another widely used method, mainly for thermal and flow problems, is the Finite Volume Method (FVM). In this method, the equations are approximated by conservation balances of the property involved (enthalpy, mass, etc.) in the elementary volume, allowing a greater association of physical interpretation with mathematics (Maliska, 1995).

Finally, it is known that the Brazilian aerospace program is at a level of development considerably lower than the programs of the United States, European Union and China, leaders of the sector in the world (SAE, 2011). In view of this, it is observed in recent years the emergence of university teams across the country that aim to encourage and enrich the space sector in Brazil. Among them is the newly created Potiguar Rocket Design team, affiliated to the Federal University of Rio Grande do Norte, whose technical problems motivated the development of this work.

## 1.1 The problem

One of the main projects developed by the aforementioned team is a 1000 m maximum apogee academic rocket. This rocket, whose model is shown in Figure 1, has its propulsion made from a class J rocket engine (approximately 1250 Ns of total impulse).

**Recovery System**    **Avionics System**    **Antenna Compartment**    **Propulsion System**
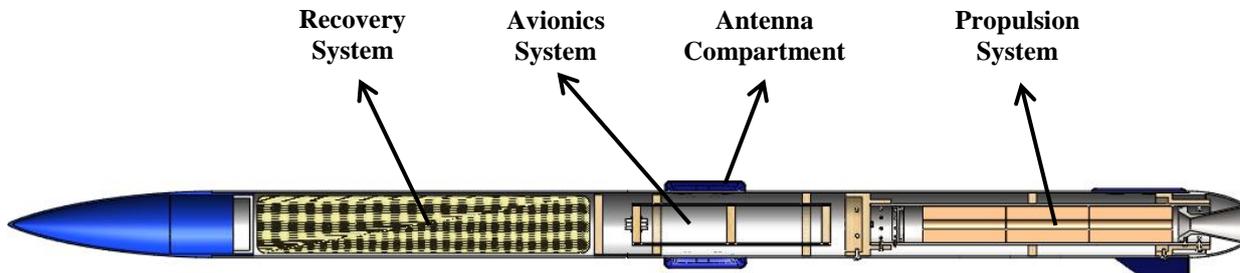
Figure 1. Academic rocket of apogee 1000 meters.

For this rocket, the material selected for the fuselage was a composite made up of 30% fiberglass mat and 70% Orthophthalic Polyester Resin. Such material was chosen because it guarantees good mechanical resistance associated with a low density and, therefore, small mass. Further, fiber and resin types were chosen from the ease to acquire and lower cost criteria. However, this material presents a problem: the maximum temperature at which the resin can be subjected before losing the structural function, known as Glass Transition Temperature (TG). For the resin used, this temperature is estimated to be at most 80°C. Thus, it can be inferred that the lower region of the rocket, where the coupled rocket engine is located, is the most critical from a thermal point of view. This region is highlighted in Figure 2.

**Bulkhead**    **Combustion Chamber**    **Centering and Fixing Rings**    **Propellant Cell**    **Nozzle**
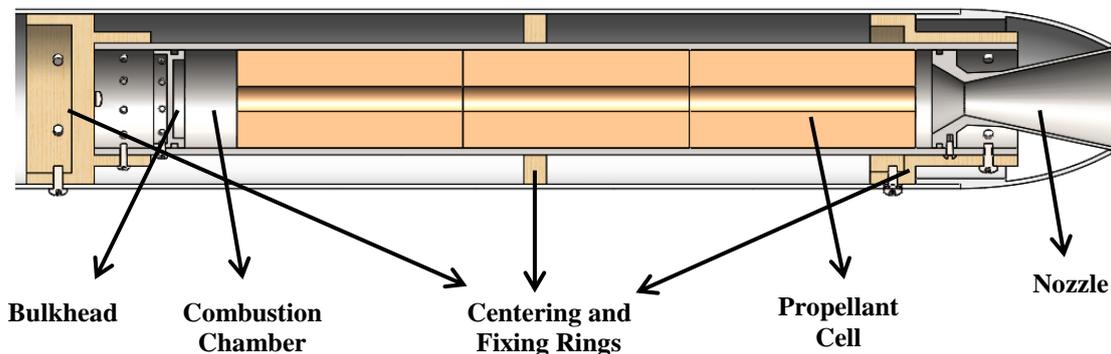
Figure 2. Propulsion system of an academic rocket.

In short, what happens is the burning of the propellant in a short period. The products of this combustion, in the gas form, are accelerated by the geometry of the nozzle and then generate thrust. It is experimentally observed that, immediately after this firing, the temperature inside the combustion chamber reaches values close to 600°C, which generates a temperature gradient accentuated with respect to the environment and facilitates the transfer of heat to the parts of the fuselage through the three existing mechanisms for such: conduction, convection and radiation (Incropera et al., 2011).

As stated, this heat transfer must be such that the body tube temperature reaches a maximum of 80°C. Once this value is exceeded, the fuselage loses its integrity and, consequently, the rocket loses stability during flight. Thus, the objective of this work was to use numerical methods to analyze the heat transference between the combustion chamber of the rocket engine and the fuselage parts made in composite (Glass Fiber + Polyester Resin), especially the body tube. The analysis was made for different insulation materials, in order to determine which is the most efficient in maintaining structural integrity.

## 2. COMPUTACIONAL PROCEDURE

### 2.1 Analysis model

Due to the symmetry of the problem, the presented system could be simplified for the model presented in Figure 3, which considers the heat flow only in the radial direction. This model is composed of three materials, named in the

figure of A, B and C. It was initially chosen not to take into account the screws present in the system, so as to focus only on the analysis of the possible insulators to be used.
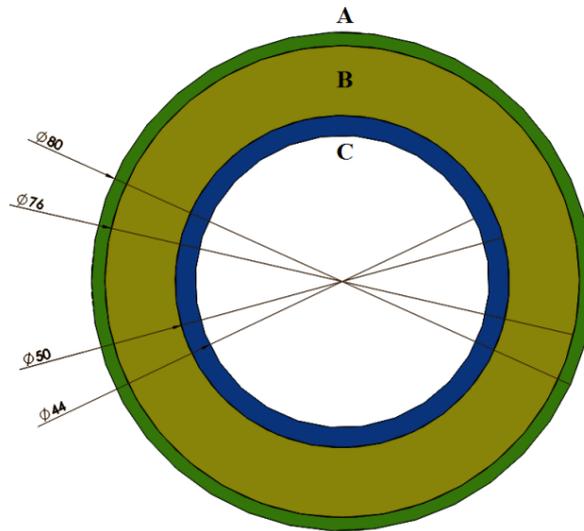


Figure 3. Simplified model of the heat transfer system. Dimension in millimeters.

To avoid heat transfer by convection and radiation, which are considerably more unpredictable processes than conduction, it was considered that this empty space between the fuselage and the engine should be completely filled. This simplifies the analysis and makes it more reliable. Thus, in the figure, A indicates the tube of composite material, C is the steel tube 1020 of the engine and B is the insulation material, which, depending on the cross-section of the rocket, needs to be a material with a structural function.

**2.2 Materials**

Table 1 presents the thermal properties (conductivity, density and specific heat) of the fuselage and engine materials, which are already defined.

Table 1. Materials already defined.

| Property | Composite (Fiberglass + orthophthalic polyester resin) | 1020 Steel alloy |
|---|---|---|
| Conductivity (W/m.K) | 0,509 | 51,9 |
| Density (kg/m³) | 1649 | 7850 |
| Specific Heat (J/kg.K) | 813,5 | 486 |

For the insulation material (B), some options have been chosen. The insulating materials were divided into two categories: the structural function materials, to be used in the engine fixing rings, and the insulating materials, which will occupy the gap between A and C and, therefore, only have the function of isolating and have as a requirement to be light, so as not to impair the performance of the rocket. These materials are presented in Table 2, together with their properties necessary for simulation of heat transfer.

Table 2. Possible materials to be used.

| Property | Structural materials | | | | Insulating materials | | | |
|---|---|---|---|---|---|---|---|---|
| | 1020 Steel Alloy | 2024 Aluminum alloy | Timber (red oak) | Technyl (nylon 6,6) | Mineral Wool | Fiber glass | Vermiculite | Expanded Clay |
| Conductivity (W/m.K) | 51,9 | 190 | 0,18 | 0,24 | 0,037 | 0,039 | 0,068 | 0,16 |
| Density (kg/m³) | 7850 | 2770 | 640 | 1140 | 85 | 57 | 80 | 400 |
| Specific Heat (J/kg.K) | 486 | 875 | 2900 | 1670 | 840 | 837 | 835 | 800 |

### 2.3 Boundary conditions

For this analysis, two boundary conditions were adopted. Are they:

- On the inner surface of the system, ie, the inside diameter of C, the temperature is set at 600 °C. This is a conservative consideration, since 600 °C is the maximum temperature at which the combustion chamber is estimated to arrive during combustion, and which occurs briefly before the temperature falls again. As the exact temperature behavior is not known, the greater value has been chosen, so that materials that isolate enough will certainly be effective in practice, even with this consideration.

- The outer surface of the system, ie, the outside diameter of A, will lose heat by convection. The value of the convective coefficient was calculated considering forced convection of the air on a cylinder, with air velocity equal to 5 m/s, which is the projected velocity with which the rocket must fall with the parachute open. Table 3 presents all the necessary information to calculate the convective coefficient, which includes air properties, cylinder diameter and flow velocity. For the calculation of Reynolds number (Re), Nusselt number (Nu) and convective coefficient (h), Equations 1, 2 and 3 were used, respectively.

Table 3. Properties for calculation of the convective coefficient. (Çengel, 2012)

| **Fluid** | Air |
|---|---|
| **Surface temperature (T)** | 25 °C |
| **Density ($\rho$)** | 1,164 kg/m³ |
| **Specific Heat ($C_p$)** | 1007 J/kg.K |
| **Thermal conductivity (k)** | 0,02588 W/m.K |
| **Thermal diffusivity ($\alpha$)** | 2,21E-05 m²/s |
| **Dynamic viscosity ($\upsilon$)** | 0,00001872 Pa.s |
| **Kinematic viscosity ($\upsilon$)** | 0,00001608 m²/s |
| **Prandtl Number (Pr)** | 0,7283 |
| **Diameter (D)** | 0,08 m |
| **Velocity (v)** | 5 m/s |

$$\text{Re} = \frac{v\,D}{\upsilon} \tag{1}$$

$$\text{Nu} = 0.193\,\text{Re}^{0,618}\,\text{Pr}^{1/3} \tag{2}$$

$$h = \frac{k\,\text{Nu}}{D} \tag{3}$$

For this work, contact resistance was despise. It is a simplification that makes the analysis more conservative and, therefore, does not affect the final decision of the insulation materials.

### 2.4 Initial conditions

Considerou-se como condição inicial as condições ambientes (temperatura igual a 25°C e pressão de 1 bar).

### 2.5 Software

O software utilizado para a simulação é o CFD Studio (Pieritz et al., 2004), desenvolvido pelo Laboratório de Simulação Numérica em Mecânica dos Fluidos e Transferência de Calor (SINMEC) da Universidade Federal de Santa Catarina (UFSC). Esse software utiliza o Método dos Volumes Finitos para fazer análises em regime transiente, tanto de condução quanto de escoamento de fluido. Este trabalho, entretanto, se limitou às análises de condução sem escoamento.

**2.6  Simulation parameters**

As previously mentioned, the analyzes of this work sought to define a material that ensures that the temperature in the composite tube never reaches the limit temperature of 80 °C, which corresponds to the glass transition temperature. It is not necessary, however, that this condition be satisfied for a long time. Adopting as maximum 5 minutes the maximum flight time (understood from launch to parachute landing), only during this time period structural integrity must be maintained. Thus, the first transient analyzes had as simulation time the 300 seconds, for which the temperature profiles, from the internal diameter to the external one, were found.

For those materials that did not reach the steady state at that time, another simulation was performed, up the steady state, using 86400 seconds (24 hours) as simulation time. In this case, in addition to the temperature profiles, the time required to achieve equilibrium was obtained.

Table 4 presents the simulation parameters for these two cases.

Table 4. Simulation parameters

| Simulation parameters | 5 minutes | Steady State |
|---|---|---|
| Increment in time (s) | 0,1 | 1 |
| Total time | 300 | 86400 |
| Number of increments | 3000 | 86400 |
| Interpolation scheme | CDS | CDS |
| Simulation tolerance | 1,00E-10 | 1,00E-10 |
| Solver type | TDMA | TDMA |
| Solver tolerance | 1,00E-05 | 1,00E-05 |
| Maximum number of solver iterations | 1000 | 1000 |
| Interval of iterations to check tolerances | 5 | 5 |

**2.7  Mesh**

To model the problem, a mesh with polar coordinates was generated. Figure 4 shows the mesh generation interface of the CFD Studio along with the I and J directions.
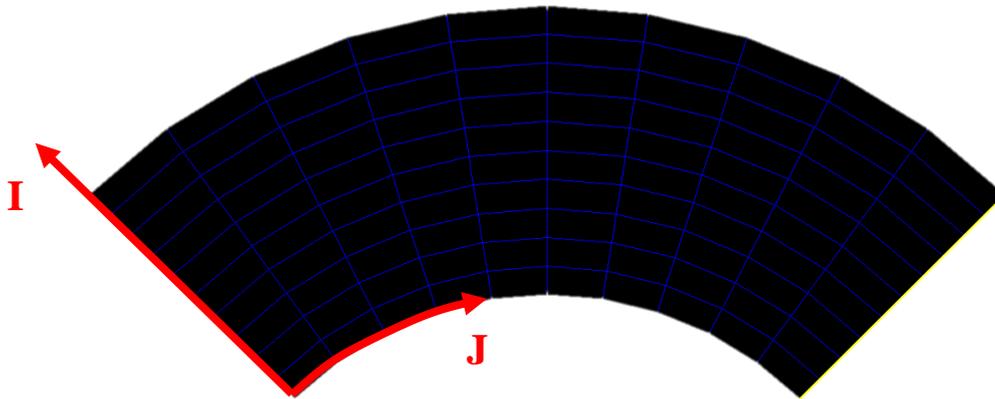


Figure 4. Mesh generated by CFD Studio.

Since, due to the characteristic of the problem, conduction only occurs in the radial direction, it was decided to perform the simulation in only 1/4 of the circle to decrease the computational effort. For this, the heat flux by the lateral limits was considered equal to zero, in order not to interfere in the results. Another consequence is that the number of volumes in the J direction little influences the results, so that a total of 25 volumes were established.

**2.7.1    Convergence test**

To determine the number of volumes in the I direction, a convergence test was performed. For this, the properties of the timber were assigned to all volumes, was used a total time of 300 s with step of 0.1 s and then the simulation was performed for different quantities of I-volumes. The first iteration was carried out with 10 volumes and, from this, the new value was defined by the multiplication of the old one by a refining ratio. For each iteration, the lowest temperature

was registered. The criterion for convergence was the temperature value error in relation to the previous iteration of is less than 0.1%.

### 2.7.2    Results validation

In order to validate the obtained results, we used the student version of the commercial software Autodesk CFD. Its analysis is based on the Finite Element Method, and allows, like the CFD Studio, the simulation of the transient conduction. This software works with 3D models and, therefore, to make the comparison, three pieces with the same diameters of the previous analysis, but with a length of 10 mm, were modeled, as can be seen in Figure 5a. Figure 5b shows the mesh model generated by this software.
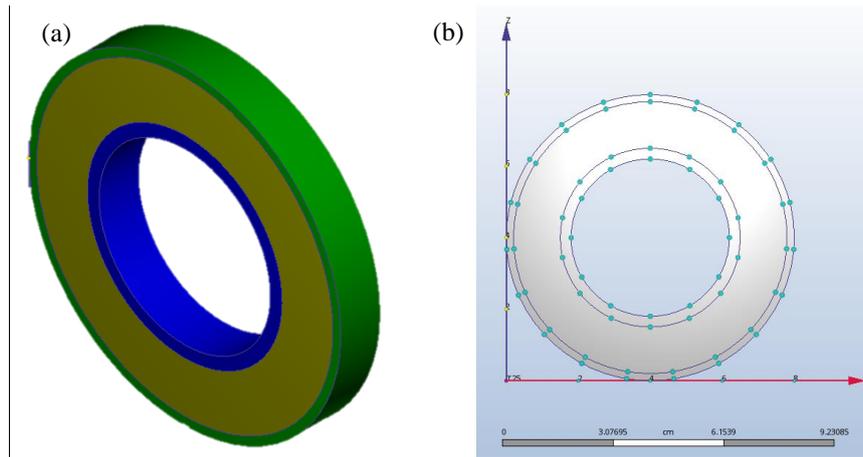


Figure 5. Autodesk CFD analysis. (a) Model created. (b) Example of mesh generated.

## 3.    RESULTS AND DISCUSSIONS

### 3.1    Convective coefficient

Based on the data in Table 3, the following values were obtained for the Reynolds number, Nusselt number and, finally, the convective coefficient h.
- Reynolds = $2,49 \times 10^4$
- Nusselt = 90,42
- h = 29,25 W/m².K

### 3.2    Convergence test

Table 5 presents the results of the iterations. It should be noted that, although the test converged at iteration 7, the mesh was more refined so that the number of volumes were multiple of the wall thickness value (18 mm), in order to facilitate the interpretation of the results.

Table 5. Iterations of the convergence test.

| Iteration | Refining ratio | I | J | I x J | Lowest temperature (°C) | Error (%) |
|---|---|---|---|---|---|---|
| 1 | - | 10 | 25 | 250 | 36,7613 | - |
| 2 | 2,00 | 20 | 25 | 500 | 36,3731 | -1,056 |
| 3 | 2,00 | 40 | 25 | 1000 | 36,0769 | -0,814 |
| 4 | 2,00 | 80 | 25 | 2000 | 35,7076 | -1,024 |
| 5 | 2,00 | 160 | 25 | 4000 | 35,4013 | -0,858 |
| 6 | 1,25 | 200 | 25 | 5000 | 35,343 | -0,165 |
| 7 | 1,20 | 240 | 25 | 6000 | 35,3093 | -0,095 |
| 8 | 1,17 | 280 | 25 | 7000 | 35,2859 | -0,066 |
| 9 | 1,14 | 320 | 25 | 8000 | 35,2737 | -0,035 |
| 10 | 1,13 | 360 | 25 | 9000 | 35,2650 | -0,025 |

Thus, were used 360 volumes in the I direction.

## 3.3    Graphical representation

Both of the software used generates, as a result of the simulation, a graphical representation. For the analysis proposed by this paper, the results display is more interesting in the graph of the Temperature x Distance from the internal surface. Thus, for example only, Figure 6 presents the graphical representations generated by the CFD Studio (Figure 6a) and by Autodesk CFD (Figure 6b).
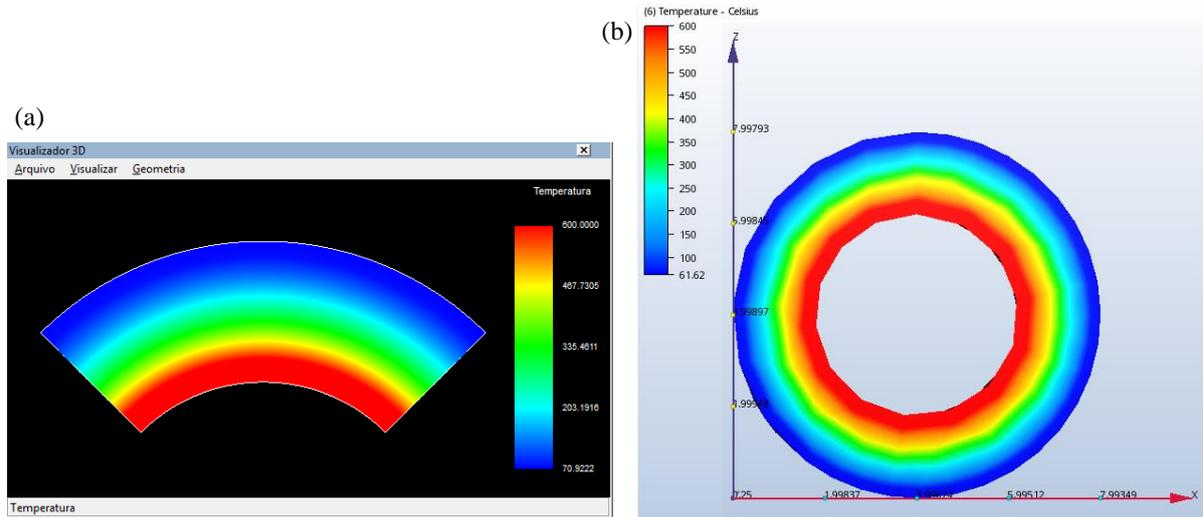


Figure 6. Graphical representation. (a) CFD Studio. (b) Autodesk CFD.

## 3.4  Temperature distribution

Figure 7 shows, for all configurations, the temperature distribution after 300 seconds of simulation. The distribution generated by materials that entered steady state before 5 minutes were not plotted.
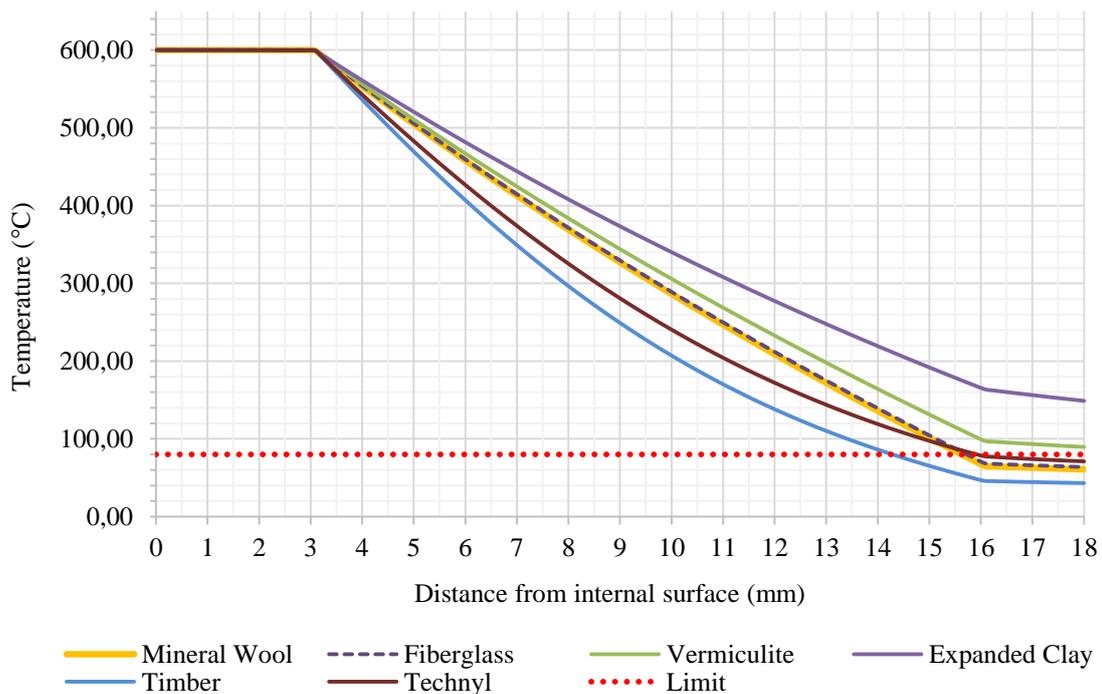


Figure 7. Temperature distribution for the simulation time of 300 seconds.

Figure 8 shows the temperature distribution for the simulations until the steady state. In this case, all settings have their profiles plotted.
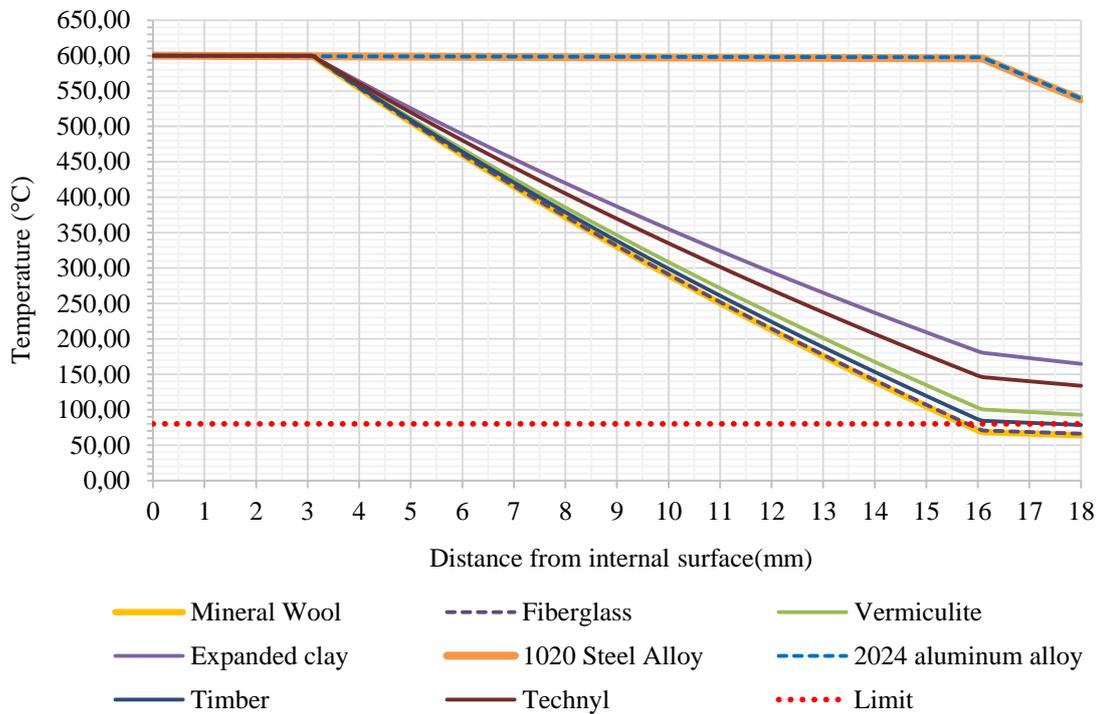


Figure 8. Temperature distribution for simulation up to the steady state.

Complementing the results, Table 6 shows the times in which each material configuration reached the steady state.

Table 6. Time up to steady state.

|  | 1020 Steel Alloy | 2024 Aluminum alloy | Timber (red oak) | Technyl (nylon 6,6) | Mineral Wool | Fiber glass | Vermiculite | Expanded Clay |
|---|---|---|---|---|---|---|---|---|
| Time up to steady state (s) | 225 | 103,9 | 4380 | 4554 | 1666 | 1557 | 1565 | 2084 |

Analyzing Figures 7 and 8, a behavior close to the linear one is observed in some distributions. Since the characteristic curve of temperature drop for models in polar coordinates usually has exponential behavior, it must be emphasized that this apparent linearity is due to the small dimensions of the model analyzed.

Thus, in the case of insulation-only materials, Fiberglass and Mineral Wool reached the objective, ensuring a temperature of less than 80 °C in the composite tube after 5 minutes. Besides, these materials proved to be viable even when analyzed until the steady state. Thus, a secondary criterion for material selection is required. Since the mass of a rocket is a crucial criterion, so that lighter rockets perform better and can carry more payload, preference will be given to the lower density material. As shown in Table 2, Fiberglass has a lower density and, therefore, is the most appropriate insulator to fill void spaces of the propulsion system.

As for the structural function materials to be adopted in the rocket engine fixing rings, the Timber and Technyl were successful in the first 5 minutes, but only the Timber proved to be able to guarantee the integrity of the composite in the steady state. In addition, the Timber is also less dense than the Technyl, so that it becomes the best choice for the construction of the rings.

### 3.5 Validation

Since the purpose of the simulations performed in Autodesk CFD is only to validate the previously presented results, the analysis for materials of similar behavior was unnecessary. Thus, up to 300 seconds and up to the steady state only the Fiberglass, the Expanded Clay and the 1020 Steel Alloy were simulated. Figures 9 and 10 show the results obtained, respectively.
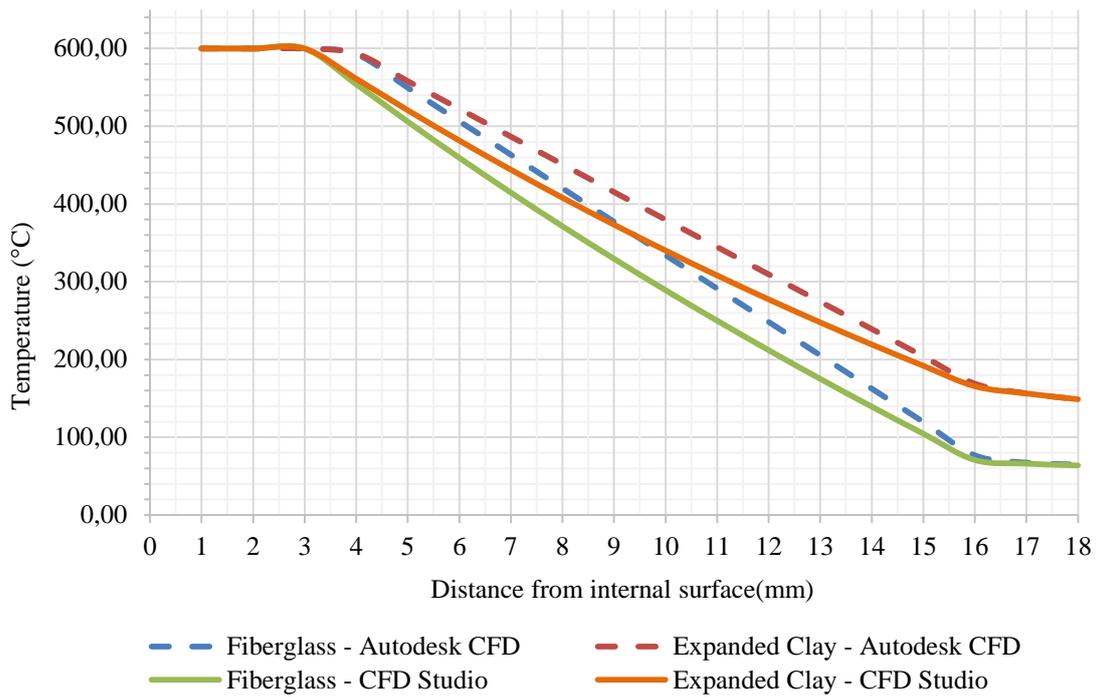
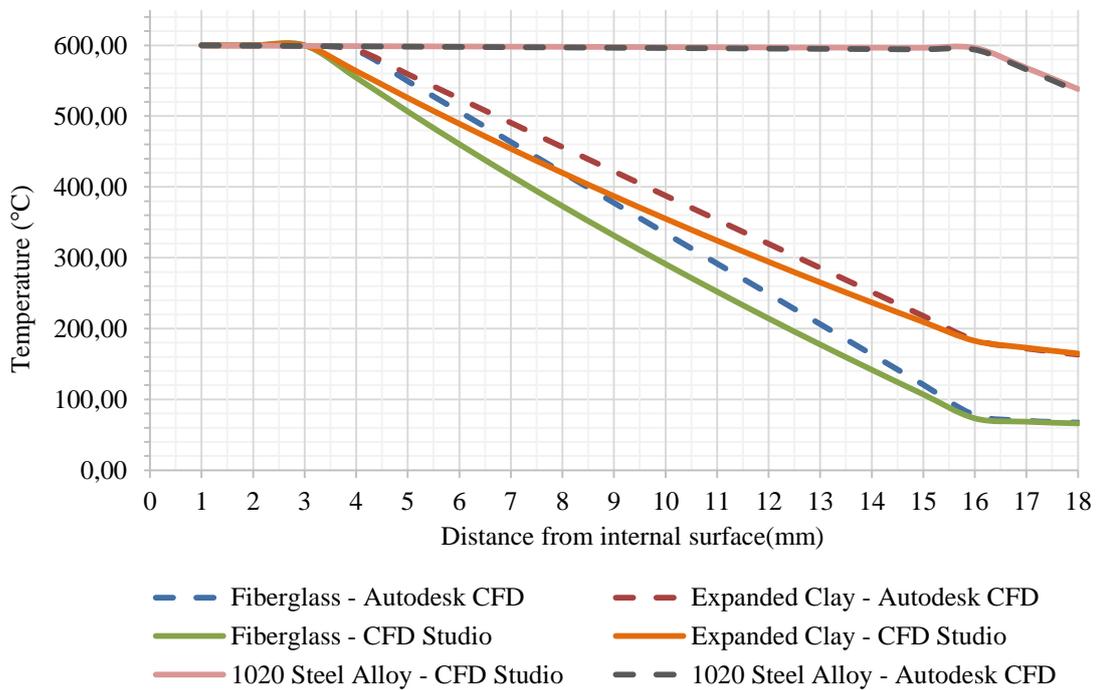Figure 9. Comparison of results. Simulation up to 300 seconds.



Figure 10. Results comparison of results. Simulation up to steady state.

Thus, although the distributions generated by CFD Studio and Autodesk CFD are somewhat different in the intermediate positions of the model, both converge to very close values of temperature at the part corresponding to the fiber tube.

## 4. CONCLUSION

Based on the results obtained, following the methodology established in this paper, it can be concluded that:

1. The application of numerical methods for the thermal analysis of the region where the rocket propulsion system is located was successful.
2. The results obtained could be properly validated.
3. The best insulation configuration to ensure the structural integrity of the rocket was: Timber for the rocket engine fixing rings and Fiberglass to fill void spaces.

## 5. ACKNOWLEDGEMENTS

## 6.  REFERENCES

Miranda, I. F. and Pessoa Filho, J. B., 2001. "Fluxo de calor convectivo sobre veículos espaciais movendo-se a velocidades hipersônicas". In *Proceedings of the 16th Brazilian Congress Of Mechanical Engineering*. Uberlândia, Brazil.

Çengel, Y. A.; Ghajar, A. J. Transferência de Calor e Massa: Uma Abordagem Prática. 4. ed. A: Amgh, 2012. 906 p.

Costa, L. E. V. L. and Moraes Jr., P., 2001. "Concepção de sistemas de proteção térmica para o veículo espacial recuperável SARA". In *Proceedings of the 16th Brazilian Congress Of Mechanical Engineering*. Uberlândia, Brazil.

Guo, Y., Liang, G., Qiu, Z. and Liu, A., 2006. "Study of a liquid insulation for the solid rocket motor". *Materials Letters*, Vol. 61, n. 11-12, p.2406-2411.

Chandrupatla, T. R. and Belegundu, A. D., 2011. *Introduction to Finite Elements in Engineering*. Pearson, New Jersey, 4 ed., 448 p.

Maliska, C. R. Transferência de Calor e Mecânica dos Fluidos Computacional. Rio de Janeiro: Ltc, 1995. 424 p.

Pieritz, R. A.; Mendes, R.; da Silva, R. F. A. F.; Maliska, C. R. CFD studio: An educational software package for CFD analysis and design. Computer Applications in Engineering Education, Estados Unidos, v. 12, p. 20-30, 2004.

SAE – Secretaria de Assuntos Estratégicos, 2011. *Desafios do Programa Espacial Brasileiro*. SAE, Brasília, 276p.

Incropera, F. P., Dewitt, D. P., Bergman, T. L., Lavine, A. S., 2011. *Fundamentals of heat and mass transfer*. John Wiley & Sons. 7ed. USA. 978-0470-50197-9

## 7. RESPONSIBILITY NOTICE

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