26th ABCM International Congress of Mechanical Engineering
November 22-26, 2021. Florianópolis, SC, Brazil

# COB-2021-0092
# THERMAL NUMERICAL ANALYSIS OF AN AIRCRAFT WING USING AN UNSTRUCTURED MESH IN CUDA-C

**Durval Marques de Queiroz Neto**
**Elisan dos Santos Magalhães**
Instituto Tecnologico de Aeronautica
Praça Marechal Eduardo Gomes, 50 - Vila das Acacias, São José dos Campos - SP, 12228-900
durval.neto@ga.ita.br
elisan@ita.br

***Abstract.*** *The ice formation problem on aircraft wings is a primary safety concern for manufacturers. Thus, it is crucial to know the temperature profile in the wing section. This work presents a thermal numerical analysis performed on an aircraft wing using parallel programming on a GPU (Graphics Processing Unit). The model evaluates the heat exchange with the environment and the temperature profile within the wing by solving the heat diffusion equation. The model considered the following boundary conditions: the prescribed temperature at the external surface and an imposed heat flux at the internal surface. The prescribed heat input condition simulates an ice protecting system commonly found in aircraft. An in-house CUDA-C code was developed to solve the two-dimensional heat transfer problem using a finite volume approach for the aircraft wing. To account for the wing profile, the model applied unstructured mesh. An independence mesh test was conducted to verify the solution. The results have shown the temperature profile for the wing tank. The results have shown that the GPU effectively performs the solution in almost real-time for a refined mesh. The main advantage of running the model in parallel within the GPU is the higher number of cores processing the problem, reducing the computational time associated cost needed to solve the problem and allowing more complex problems to be solved in commercial computers instead of clusters.*

***Keywords:*** *CUDA, heat flux, aircraft wing, parallel processing, heat conduction*

## 1. INTRODUCTION

Heat transfer problems are widely studied due to their importance to engineering applications. The simplest technics to study heat transfer problems involves approximation of the geometry with orthogonal coordinating systems. This approximation allows the simplification of how the governing equations are written and processed by the solver algorithm. However, most engineering problems involve complex geometries that cannot be simplified to orthogonal coordinating systems without inducing high errors.

For complex geometry, the CFD programs use the unstructured grid. The main advantage of an unstructured grid is that each cell is treated as a block. Thus, the mesh concentrates where it is necessary. This approach gives almost unlimited geometry flexibility without spending a long time on mesh generation and mapping. However, there might be neighborhood mesh elements with non-orthogonality between the elements' centroids and the nodes' surfaces for an unstructured grid. This characteristic is known as mesh skewness or non-orthogonality. For these elements, the flux calculation has to be corrected by adding a contribution caused by the non-orthogonality.

Mathur and Murthy (1997) proposed a method to calculate the correction caused by the non-orthogonality of the diffusive term in unstructured cell-centered grid mesh. They also presented the gradients required for the evaluation of diffusion fluxes. The method proposed by them showed a good correlation when compared with problems found in the literature. The method proposed by Mathur and Murthy (1997) was also used in Versteeg and Malalasekera (2007).

Guixia et al.(2008) also presented an approach for approximating the energy flux of temperature diffusion equation on unstructured meshes based on the two-dimensional Finite Volume Method. Their approach can be applied to unstructured meshes, such as arbitrary polygons and non-matched meshes.

Lyra et al. (2004) developed a numerical analysis of heat conduction applications in unstructured meshes considering a tri-dimensional model, which was also simplified for the two-dimensional model. The authors developed an unstructured finite volume vertex-centered formulation, which was implemented using an edge-based data structure. The model results were compared with simple heat transfer problems, showing good accuracy.

A study to evaluate the comparison of CPU and GPU performance for solving the convection-diffusion equation using a modified SOR method was conducted by Cotronis et al. (2014). The modified SOR method showed that the GPU reached a speedup factor of more than eight over the single-threaded CPU version.

Knowing the temperature profile in an aircraft wing is essential due to icing formation in the wing leading edge. Alekseenko and Prikhod'ko (2014) developed a mathematical model to evaluate the icing of aerodynamic surfaces. The authors considered the icing meteorological parameters, the cloud types, the types of ice coating, and the protection systems against icing for the analysis. The authors also evaluate how the ice formed in the wing leading edge affected the airflow through the wing.

Experimental correlations of heat transfer at different angles of attack using a NACA 63-421 airfoil profile were proposed by Wang et al. (2008). The authors obtained empirical heat transfer coefficients for different Reynolds numbers varying the attack's angles from 0 to 25 deg. In addition, the paper showed that the experimental data could be collapsed onto a normalized correlation varying Prandtl numbers for steady-state conditions.

This paper performs a thermal analysis over the NACA 63-421 airfoil profile using a modified SOR method using a vertex-centered unstructured grid. The solver algorithm was written to run parallelized on a GPU in the CUDA-C programming language. The main advantage of using the unstructured grid is to deal with the complex airfoil geometry. Nevertheless, as the complexity of the problem increases, the computational time also increases. However, the computational time was reduced due to the parallel solving algorithm, which can be solved almost in real-time. The results were compared with those obtained during tests performed by Wang et al. (2008) to validate the proposed methodology.

## 2. MODEL DEVELOPMENT

### 2.1 Theoretical formulation

The governing equation to evaluate the heat diffusion problems for the 3D cartesian and orthogonal coordinates, extracted from (Bergman et al. 2014) is:

$$\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)+\dot{g}=\rho c_p\frac{\partial T}{\partial t} \tag{1}$$

Where $k$ is the thermal conductivity, $T$ is the temperature, $t$ is the time, $\rho$ is the specific mass, $\dot{g}$ is the generation source, $c_p$ is the specific heat, and $x$, $y$, and $z$ are the orthogonal coordinate distances.

This paper considered a 2D steady-state geometry without a heat generation source. The thermal conductivity was considered constant within the temperature range in this analysis. The governing equation (1) can be rewritten considering these assumptions as:

$$\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)=0 \tag{2}$$

Equation (2) can be discretized in a finite volume domain using an unstructured mesh grid. The general discretized equation for the boundaries of the mesh grid considering the correction caused by the non-orthogonality can be written as presented by Mathur and Murthy (1997):

$$\begin{aligned}\sum_{All\_surfaces}\left(\int_0^{\Delta A_i}\mathbf{n}_i(k\nabla T)dA\right)&=\sum_{All\_surfaces}\left(\frac{k}{\Delta\xi}\frac{\mathbf{n}\cdot\mathbf{n}}{\mathbf{n}\cdot\mathbf{e}_\xi}\Delta A_i(T_A-T_P)-k\frac{\mathbf{e}_\xi\cdot\mathbf{e}_\eta}{\mathbf{n}\cdot\mathbf{e}_\xi}\frac{\Delta A_i}{\Delta\eta}(T_b-T_a)\right)\\&=\sum_{All\_surfaces}\left(D_i(T_A-T_P)-S_i\right)\end{aligned} \tag{3}$$

where $\Delta\xi$ is the distance between the centroid of the element $P$ and the centroid of the adjacent element, $n$ is the normal vector orthogonal to the element surface, $e_\xi$ is the normal vector between the centroid of the element $P$ and the centroid of the adjacent element, $e_\eta$ is the normal vectors parallel to the element surface, $T_p$ and $T_A$ are the temperatures of the element $P$ and adjacent element $A$, $T_a$ and $T_b$ are the temperatures of the points $a$ and $b$, $\Delta\eta$ is the area of the surface of the element, and $\Delta A_i$ is the surface of the area between the element $P$ and the adjacent element $A$. For more details, see Figure 1.

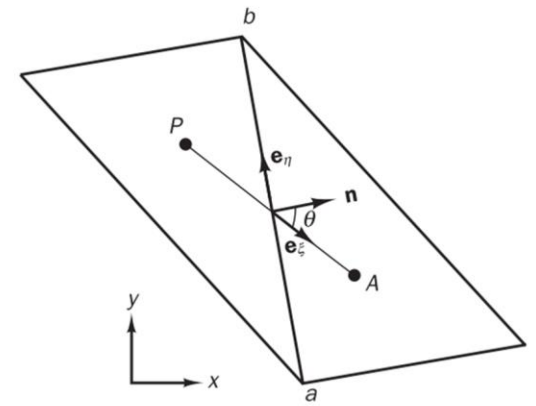


Figure 1. Schematic view of the finite element volume.
(Source: Versteeg and Malalasekera (2007))

The equation (2) means that as the analysis is performed at steady-state conditions, the sum of the energy balance is equal to 0. For this analysis, the adopted mesh is triangular, meaning that the maximum number of adjacent nodes for each element is 3. Therefore, the equation (3) can be rewritten as:

$$T_P D_P = T_A D_A + T_B D_B + T_C D_C + S \tag{4}$$

where $D_p$ is equal to the sum of all the direct terms related to the element ($D_P = D_A + D_B + D_C$) and $S$ is the sum of all the cross-diffusive terms that affect the element $P$, to calculate the cross-diffusion term the temperature at the points $a$ and $b$. These points are assumed to be an average of the temperature of the neighborhood elements.

### 2.2 Model description and boundary conditions

This analysis considers two boundary conditions: the prescribed temperature on the external surface and heat flux imposed at the internal surface.

The temperature imposed at the external surface is obtained using the empirical correlation for the average Nusselt number proposed by Wang et al. (2008). For this analysis, the temperature is considered constant throughout the airfoil.

The heat flux boundary condition is imposed on the internal surface to model the Icing protecting system, generally found in aircraft. The icing protecting system commonly uses the aircraft engine bleed air or an electro-thermal resistance to heat the wing internal surface. Therefore the external surface temperature also increases and reduces the icing formation. The heat flux is considered constant throughout the internal surface.

For the boundary nodes, the model assumed two boundary conditions: prescribed heat flux and imposed convection. The following adjustment needs to be performed in the coefficients shown in the equation (4).

For the surface which the heat flux is imposed, the equation (4) is rewritten as:

$$T_P D_P = T_A D_A + T_B D_B + S + \Delta\eta\dot{q} \tag{5}$$

$$D_P = D_A + D_B \tag{6}$$

For the surface which is subject to convection, the equation (3) is rewritten as:

$$T_P D_P = T_A D_A + T_B D_B + S - \frac{k}{\Delta\xi} T_0 \tag{7}$$

$$D_P = D_A + D_B + \frac{k}{\Delta\xi} T_0 \tag{8}$$

The regions of the airfoil subject to the boundary conditions are presented in Figure 2.

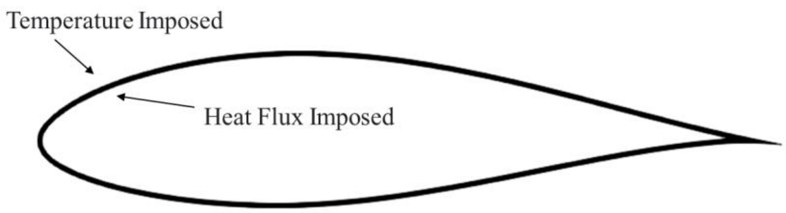


Figure 2: Boundary Conditions.

### 2.3 Program description

An in-house code was developed to solve the diffusion equation on unstructured meshes. The code was written in CUDA-C language. The code first read the mesh data and the boundary conditions data file. The mesh file was created using the open-source Gmsh software presented by Geuzaine and Remacle (2009). In this code, the data is saved in the *.su2 extension. Next, the code extracts the following information from the mesh file in *.su2 extension: The points numbering, the elements numbering and the points that compose each element, the points coordinate, the type of the boundary condition, and points located at the surface.

After loading this information, each element centroid is calculated, then the adjacent nodes for each element and point are defined. The direct term showed in the equation could be calculated with the adjacent nodes for each element defined and the geometric information for all points calculated. After that, the coefficients shown in the equation (4) are modified to introduce the boundary conditions to the applicable elements. Finally, with all the previous steps complete, the parameters are loaded into the solver. In this analysis, a modified version of SOR is used as the solver. For the convergence criteria, the residual temperature shall be less than $1.0\times10^{-6}$. When the temperature convergence is complete, the temperature field is saved in *.vtk file format, loaded to the visualization software. The coding algorithm is presented in Figure 3. It also shows if each step of the code runs at the CPU or runs at the GPU.

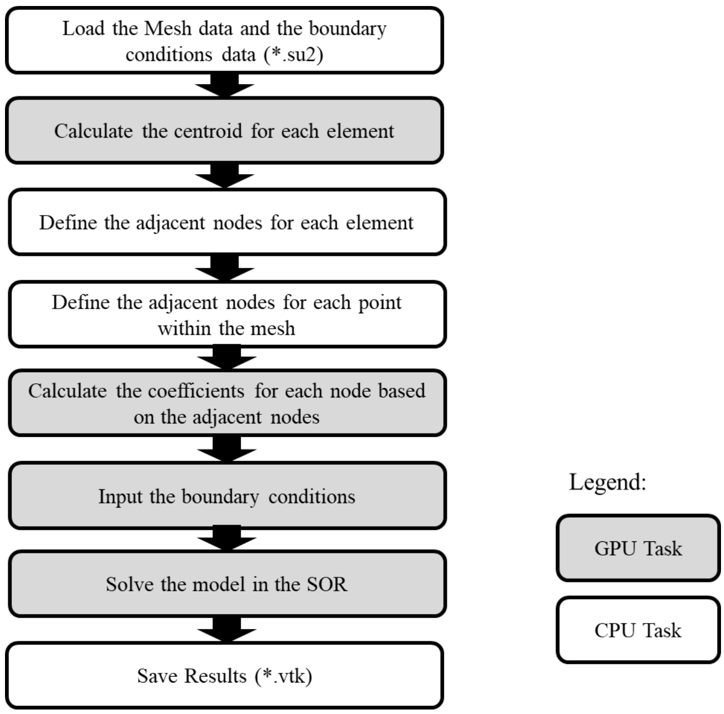


Figure 3: Code Flow down

The solver used in this analysis is a modified version of the SOR (Successive Over-Relaxation Hadjidimos (2000)) method. First, for each element, the mesh vectors were arranged to create a diagonal matrix. Then, for each iteration, the cross-diffused term was evaluated.

### 2.4 Mesh Selection and Mesh Convergence Test

The mesh used in this analysis has 76622 elements. The dimensions of the airfoil are the same used in the tests conducted by Wang et al. (2008). The chord has a length of 0.5m, and the thickness of the airfoil is 3mm. Figure 4 shows the geometry and the mesh of the airfoil used in this analysis. As one can see, the unstructured mesh was able to capture the complex geometry of the airfoil.

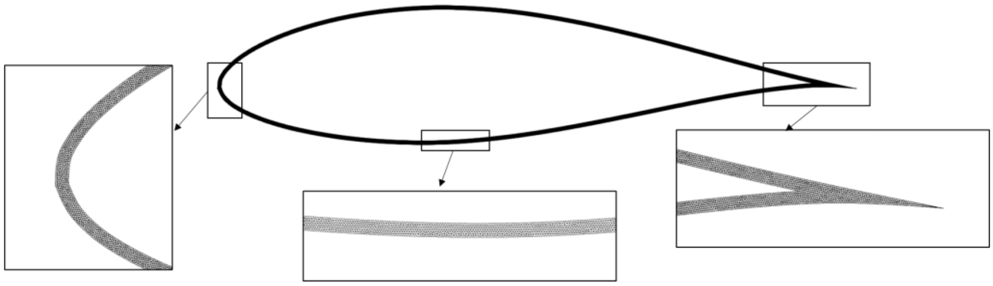

Figure 4. Used mesh for the analysis.

A mesh convergence test was performed to verify that the mesh selected for this analysis is refined enough to model the temperature profile. Table 1 summarizes the size of the mesh and the time required by the SOR solver to complete the calculation of the temperature field over the airfoil profile.

Table 1: Mesh convergence test results.

| N-ELEM [-] | Time to complete [s] |
|---|---|
| 1396 | 0.115 |
| 10576 | 0.735 |
| 21985 | 1.475 |
| 28185 | 1.789 |
| 43572 | 2.956 |
| 76622 | 5.433 |
| 169755 | 7.433 |

Figure 5 shows the temperature profile at the wing leading edge calculated using the mesh with the number of elements defined in Table 1.

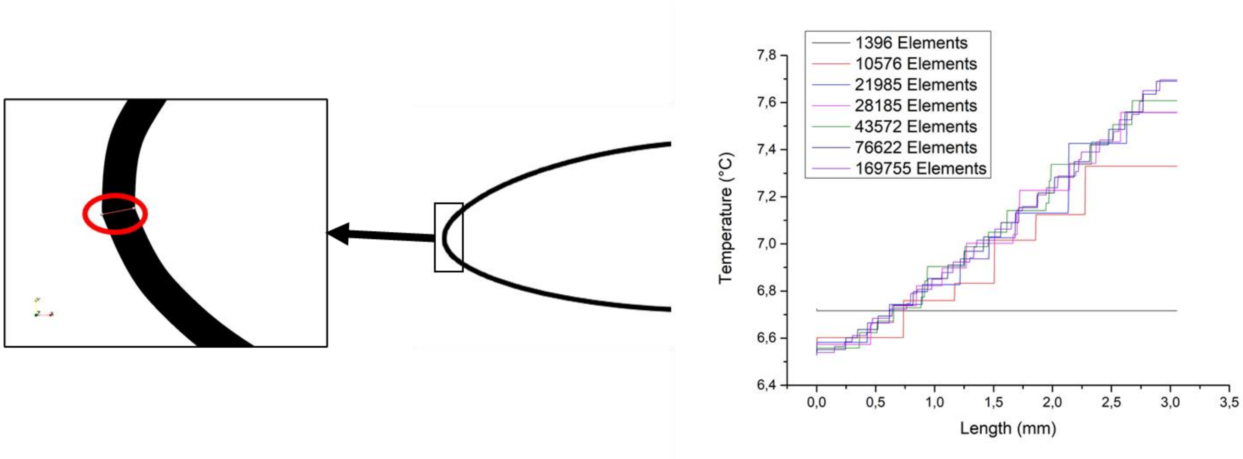


Figure 5. Mesh convergence test.

As one can see, the mesh with 76622 elements showed similar results compared with a mesh composed of 169755 elements. Therefore, it can be concluded that the mesh with 76622 elements was refined enough to produce a good temperature profile at the wing surface.

## 3. RESULTS

Figure 6 presents the temperature calculated throughout the airfoil. The leading edge, the upper surface, and the trailing edge are highlighted to show that the unstructured mesh captures the temperature field of the complex geometry. As one can see, the evaluated temperature profile is adherent with the expected results. As the inner segment is subject to positive heat flux, to model the ice protection system, and the external surface is considered to have a fixed temperature, the highest temperatures are found at the internal surface.

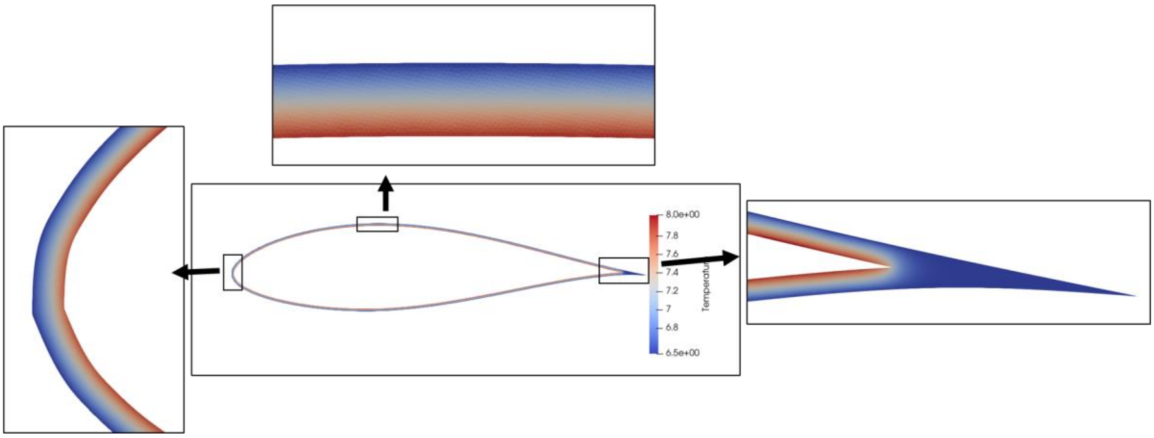


Figure 6. Temperature field at the NACA 63-421 airfoil.

Figure 7 shows three temperature profiles extracted from the results: one at the leading edge, one at the upper, and one at the bottom. The length = 0 is located at the external surface, and the length = 3mm is located at the internal surface. The difference between the maximum and the minimum temperature remained below 2°C corroborating the hypothesis that thermal conductivity can be considered constant throughout the analysis. This low-temperature difference is explained by the thickness of the plate used by Wang et al. (2008) and due to the lowest heat flux imposed at the internal surface.

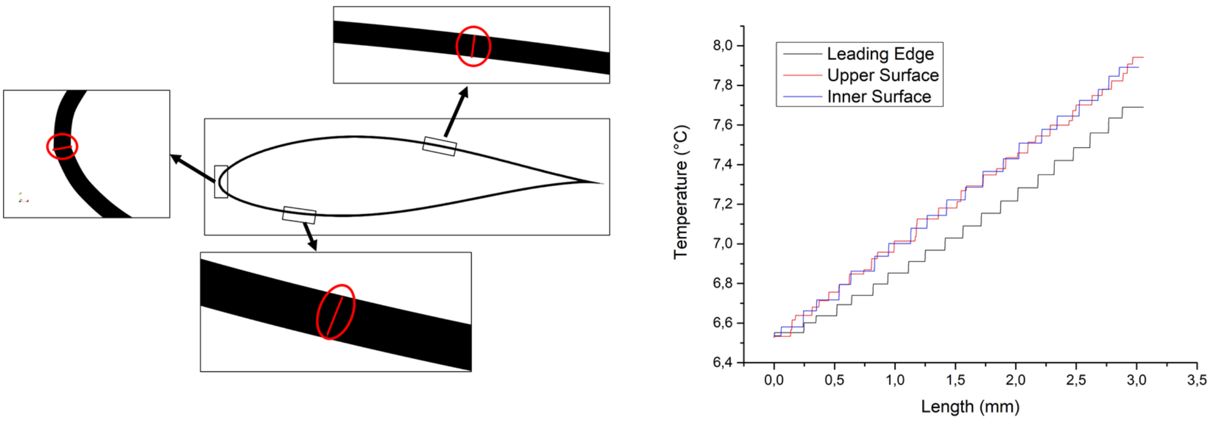


Figure 7. Temperature profile at the NACA 63-421 airfoil.

The temperature calculated throughout the airfoil at the upper surface, shown in Figure 7, by the in-house code were then compared with calculated results using the solver for the thermal steady state from the commercial software ANSYS © with 76709 elements, assuring that the solver yields correct results. Figure 8 compares the temperature calculated by the in-house code and the ANSYS © for the upper surface.

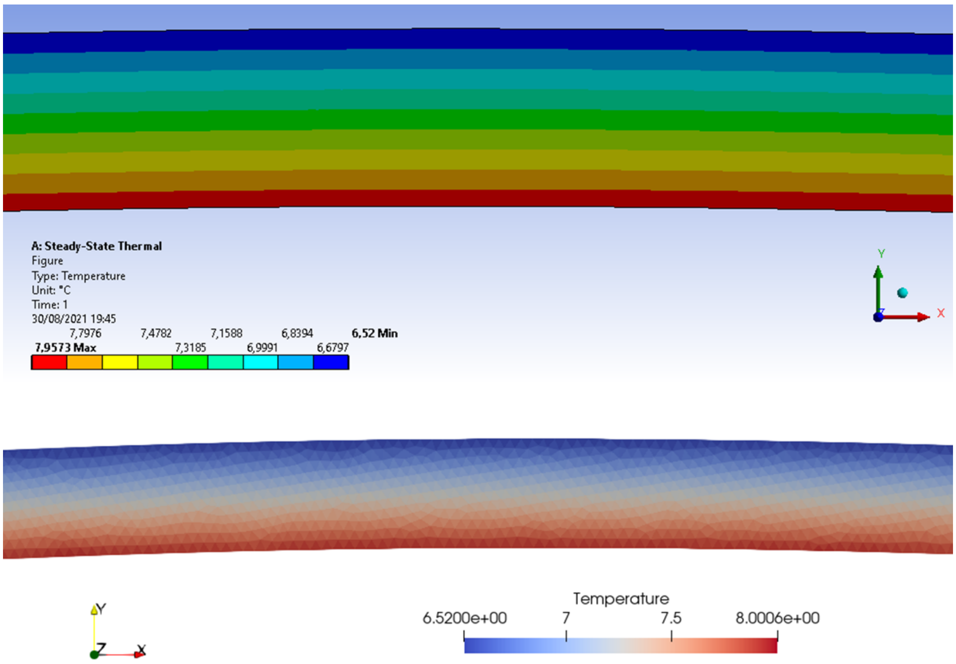


Figure 8: Temperature comparison between the in-house code and ANSYS © results.

## 4. CONCLUSIONS

Thermal analysis of a NACA 63-421 airfoil profile using an unstructured grid was shown. The code was developed to run parallelized in a GPU used a modified version of the Successive Over-Relaxation (SOR) method. A mesh convergence test was performed to ensure that the mesh covers the complex airfoil geometry. The temperature field calculated by the model captured the complexity of the airfoil geometry and was validated by comparing it with commercial software.

For the following projects, the fluid dynamics using an unstructured mesh will be developed to capture the effect of the fluid flowing through the airfoil and the airfoil's internal flow. The code will also be developed to run parallelized on a GPU.

## 5. ACKNOWLEDGEMENTS

The authors would like to acknowledge CAPES and CNPq for the financial support provided.

## 6. REFERENCES

**Alekseenko and Prikhod'ko** (**2014**) S. V. Alekseenko and A. A. Prikhod'ko. Mathematical modeling of ice body formation on the wing airfoil surface. *Fluid Dynamics*. 49, 6 (2014), 715–732. doi: 10.1134/S0015462814060039.

**Bergman et al.** (**2014**) L. T. Bergman et al. *FUNDAMENTALS OF HEAT AND MASS TRANSFER, SEVENTH EDITION*.

**Cotronis et al.** (**2014**) Yiannis Cotronis et al. A comparison of CPU and GPU implementations for solving the Convection Diffusion equation using the local Modified SOR method. *Parallel Computing*. 40, 7 (2014), 173–185. doi: 10.1016/j.parco.2014.02.002.

**Geuzaine and Remacle** (**2009**) Christophe Geuzaine and Jean François Remacle. Gmsh: A 3-D finite element mesh generator with built-in pre- and post-processing facilities. *International Journal for Numerical Methods in Engineering*. 79, 11 (2009), 1309–1331. doi: 10.1002/nme.2579.

**Guixia et al.** (**2008**) Lv Guixia et al. Numerical methods for energy flux of temperature diffusion equation on unstructured meshes. *Training*. 4179, September 2008 (2008), 53. doi: 10.1002/cnm.

**Hadjidimos** (**2000**) A. Hadjidimos. Successive overrelaxation (SOR) and related methods. *Journal of Computational and Applied Mathematics*. 123, 1–2 (2000), 177–199. doi: 10.1016/S0377-0427(00)00403-9.

**Lyra et al.** (**2004**) P. R. M. Lyra et al. An edge-based unstructured finite volume procedure for the numerical analysis of heat conduction applications. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*. 26, 2 (2004), 160–169. doi: 10.1590/S1678-58782004000200007.

**Mathur and Murthy** (**1997**) S. R. Mathur and J. Y. Murthy. A pressure-based method for unstructured meshes. *Numerical Heat Transfer*, *Part B: Fundamentals*. 31, 2 (1997), 195–215. doi: 10.1080/10407799708915105.

**Versteeg and Malalasekera** (**2007**) H. K. Versteeg and W. Malalasekera. *An Introduction to Computational Fluid Dynamics THE FINITE VOLUME METHOD*. doi: 10.24114/eb.v2i2.4393.

**Wang et al.** (**2008**) X. Wang et al. Convective heat transfer from a NACA airfoil at varying angles of attack. *Journal of Thermophysics and Heat Transfer*. 22, 3 (2008), 457–463. doi: 10.2514/1.34405.

## 7. RESPONSIBILITY NOTICE

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