XXVIII Congresso Nacional de Estudantes de Engenharia Mecânica
09 a 13 de maio de 2022, Santa Maria, Rio Grande do Sul, Brasil

# Preliminary Studies of the Influence of Surface Temperature on the Aerodynamic Coefficients of Biomimetic-based Airfoils

**Andres Gilberto Machado da Silva Benoit, andres.benoit@acad.ufsm.br**[1]
**Luca Sauer do Araujo, luca.sauer@acad.ufsm.br**[1]
**Luiz Henrique Schaffazick, luiz.schaffazick@acad.ufsm.br**[1]
**Giuliano Demarco, giuliano.demarco@ufsm.br**[2]
**César Gabriel dos Santos, cesar.g.santos@ufsm.br**[2]
[1]Curso de Engenharia Aeroespacial, Universidade Federal de Santa Maria
[1]Departamento de Engenharia Mecânica, Universidade Federal de Santa Maria

***Resumo:****Sabe-se que os aerofólios representam um importante objeto de estudo aerodinâmico e procura-se sempre melhorar seus coeficientes de sustentação e arrasto. Somado a isso, há muito tempo que a engenharia procura por soluções ótimas através da observação da natureza, tal estudo inspirado na biologia chama-se de biomimética. Diante disso, o presente trabalho tem como objetivo verificar a influência de uma diferença de temperatura entre o intra e extradorso sobre os coeficientes aerodinâmicos de um aerofólio simétrico NACA 0012, com base em um processo que pode ser encontrado em aves. Para buscar as diferenças, métodos numéricos computacionais de CFD foram utilizadas levando em consideração escoamentos de baixo número de Reynolds. Além disso, foram avaliadas as possibilidade de aquecimento em diferentes superfícies separadamente, também analisa-se a razão entre coeficiente de sustentação e coeficiente de arrasto. Para uma diferença de 50ºC, percebemos que o aquecimento na superfície inferior promove melhoras na sustentação com relação ao arrasto de escoamentos para o aerofólio simétrico para a maioria significativa dos ângulos de ataques.*

***Palavras-Chave:*** *Aquecimento, Coeficiente de Sustentação, Coeficiente de Arrasto, Aerofólio NACA0012, Razão Sustenação-Arrasto*

***Abstract.*** *It is well known that airfoils represent an important subject of aerodynamics studies and the search for better drag and lift coefficients ($c_D$ and $c_L$, respectively) is continuously ongoing. Beyond that, for a long time engineering has been searching for optimal solutions through nature observation, in a process inspired on biology and called biomimetics. Considering this, the current work is intended to ascertain the influence of a temperature difference between lower and upper surface over the aerodynamics coefficients of a symmetric airfoil NACA 0012, based on a process that can be found on birds. The study makes the evaluation of viability of implementation of such methods possible and also looks into the interactions between temperature and flow over lift devices. In order to do that, numerical methods of Computational Fluid Dynamics (CFD) were applied taking into account Low Reynolds Number. In addition to that, warming possibilities of upper and lower surface were both separately evaluated. The analysis was carried out considering the ratio between $c_L$ and $c_D$ and the results obtained show that, for a 50ºC difference, the heating of the lower surface promote improvement on the lift while maintaining the drag coefficient nearly the same, thus enhancing the lift to drag ratio for the significant majority of angles of attack analyzed.*

***Keywords:*** *Heating, Lift Coefficient, Drag Coefficient, Airfoil NACA0012, Lift-to-Drag Ratio*

## 1. Introduction

Over the last few years, several aircraft models have been developed, each with its own characteristics and designed for specific needs. An example of this is the Micro Aerial Vehicles (MAV), which are small aircraft that due to its size, agility and capability of reaching remote places, several areas of science such as: robotics; aerodynamics; mechatronics and even the military have been looking for ways to improve their use.

One of the problems found in these aircraft is directly related to the operation in the range of the Low Reynolds Number flow (Re), where the flow is separated over the airfoil, which increases the drag and reduce the lift. The purpose of this study is to improve the aerodynamics of a NACA0012 airfoil in this range of Re. On considering this issue, one of the ways to try to reduce it is through a phenomenon known as Thermal Camber, which consists of cooling the upper surface and heating lower surface, improving the aerodynamic performance.

The foundation for the use of the NACA0012 is the fact that is a well-known airfoil in researches, and the amount of published bibliography. Although, due to its symmetric geometry, just the drag coefficient undergoes significant changes (Eftekhari and Al-Obaidi, 2019; Ahmed *et al.*, 2013a).

According to Benyus (1997) and Vincent (2009), looking into biology shows us solutions in many levels and possibilities, but technology needs to close the gap to the problems trying to be solved through engineering. As Bottlender *et al.* (2021) concluded, biomimetics represent an interdisciplinary solution with emphasis on engineering functionalities. Considering this scenario, some birds in nature are found to have different temperatures over their upper and lower wing surfaces, such as albatrosses. More commonly, this difference is caused by their skin color jointly to their exposition to solar irradiation. As they have different colors, it's plausible to consider that their skin different surface color and therefore temperature during operation shall had made some contribution in aerodynamics for these animals. Regarding birds more explicitly, on Hassanaliana *et al.* (2017) some deeper studies over the flight abilities of the albatross can be found and also a thermal study considering similar factors to what is proposed on this paper.

Similar studies considering a temperature difference between lower and upper surface and the influence it brings to lift and drag coefficients have been made. It was found by Hassanaliana *et al.* (2019) that, considering low angles of attack, the efficiency can be significantly increased by the use of thermal heaters. While Samiee *et al.* (2018) discovered that the thermal camber phenomena of heating the lower surface produces enhancement of aerodynamic performance for some airfoils, however, the authors did not evaluate a range of angles of attack.In this work, we evaluate the temperature influence on the aerodynamic coefficients for a chosen airfoil.

## 2. Objectives

As the main objectives for this study, it can be quoted:

- Investigate the influence of a thermal difference imposed between upper an lower surfaces of an NACA 0012 airfoil aerodynamic coefficient. Both cases were considered, with the upper and lower surface being evaluated separately;

- Provide results that may help indicate if the use of heating devices on the wing of vehicles operating at Low Reynolds Number is economical and aerodynamically feasible. As these vehicles tend to be lightweight, their possible aerodynamic improvement by this temperature difference may not compensate the weight that could be added to the system in order to generate this heat.

## 3. Methodology

One of the first numerical algorithm was the root-finding method for solving simple equations in old Egyptian Rhind papyrus (1650 BC), this show the importance of numerical simulations in modern engineering since it was an important precursor to the development of calculus by Isaac Newton (1642-1727) and Gottfried Leibniz (1646-1716). In the actual days, one of main applications of numerical algorithms is in fluid mechanics, mainly known as computational fluid dynamics (CFD) and nowadays modern engineers apply both experimental and CFD analysis, and the two complement each other in fluid mechanics problems (Çengel and Cimbala, 2018). In order to look for the influence of the temperature on the aerodynamics coefficients, bi-dimensional numerical simulations were taken for several angles of attack and then the obtained values were curve-fitted with linear interpolation.

### 3.1 Numerical Simulation

The history of CFD theory started early 1970s and this was triggered by the availability of increasingly more powerful mainframe of computer technology. Among the first applications of CFD methods was the simulation of transonic flows based on the solution of the non-linear potential theory (Blazek, 2015). The current state of CFD can handle laminar flows with ease but in other hand, turbulent flows where the attention of engineering is mainly focused, are impossible to solve without invoking turbulence models and there is no universal model for turbulence, so it has to be chosen carefully appropriated to the problem (Çengel and Cimbala, 2018).

CFD algorithms solve multiple fluid dynamics situation, in general, it solves the Navier Stokes equation and other conservative and non-conservative physics laws in a mathematical language. It follows that CFD solves for a viscous flow,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0, \tag{1}$$

$$\begin{aligned}
\text{x-component}: \frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u \mathbf{V}) &= -\frac{\partial p}{\partial x} + \frac{\partial \tau_{\mathrm{xx}}}{\partial x} + \frac{\partial \tau_{\mathrm{yx}}}{\partial y} + \frac{\partial \tau_{\mathrm{zx}}}{\partial z} + \rho f_{\mathrm{x}}, \\
\text{y-component}: \frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho v \mathbf{V}) &= -\frac{\partial p}{\partial y} + \frac{\partial \tau_{\mathrm{xy}}}{\partial x} + \frac{\partial \tau_{\mathrm{yy}}}{\partial y} + \frac{\partial \tau_{\mathrm{zy}}}{\partial z} + \rho f_{\mathrm{y}}, \\
\text{z-component}: \frac{\partial (\rho w)}{\partial t} + \nabla \cdot (\rho w \mathbf{V}) &= -\frac{\partial p}{\partial z} + \frac{\partial \tau_{\mathrm{xz}}}{\partial x} + \frac{\partial \tau_{\mathrm{yz}}}{\partial y} + \frac{\partial \tau_{\mathrm{zz}}}{\partial z} + \rho f_{\mathrm{z}},
\end{aligned} \tag{2}$$

$$
\begin{aligned}
\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla\cdot\left[\rho\left(e+\frac{V^2}{2}\mathbf{V}\right)\right]
&=\rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)\\
&+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}+\frac{\partial\left(u\tau_{\mathrm{xx}}\right)}{\partial x}\\
&+\frac{\partial\left(u\tau_{\mathrm{yx}}\right)}{\partial y}+\frac{\partial\left(u\tau_{\mathrm{zx}}\right)}{\partial z}+\frac{\partial\left(v\tau_{\mathrm{xy}}\right)}{\partial x}+\frac{\partial\left(v\tau_{\mathrm{yy}}\right)}{\partial y}\\
&+\frac{\partial\left(v\tau_{\mathrm{zy}}\right)}{\partial z}+\frac{\partial\left(w\tau_{\mathrm{xz}}\right)}{\partial x}+\frac{\partial\left(w\tau_{\mathrm{yz}}\right)}{\partial y}+\frac{\partial\left(w\tau_{\mathrm{zz}}\right)}{\partial z}+\rho\mathbf{f}\cdot\mathbf{V},
\end{aligned}
\tag{3}
$$

where the equations 1, 2, 3 are known as continuity, momentum and energy conservation equations, respectively. Also, $\mathbf{V}$, $\rho$, $p$, $k$, $T$, $\tau_{ii}$, $\tau_{ij}$ ,$\dot{q}$, $e$, $\mathbf{f}$ stands for velocity vector, density, pressure, thermal conductivity, temperature, normal stress, shear stress, volumetric heat addition, specific energy and field forces (such as gravity), respectively. In addition, the velocity vector is decomposed in $x$, $y$ and $z$ components known as $u$, $v$, $w$, respectively (Anderson, 1995; Versteeg and Malalasekera, 2007; Blazek, 2015). All the numerical study is simulated using Ansys Fluent 2021 R2 Student Edition, widely used in CFD application.

### 3.2 Airfoil Geometry

A common nomenclature for airfoil geometrical parameters is shown in Fig. 1. The mean camber line is the locus of point halfway between the lower and upper surface. Then, the most forward and upward points of the mean camber line are the leading and trailing edges, respectively. The straight connection of leading and trailing edge is the chord line and the precise distance of this line is called chord, denoted by $c$ (Anderson, 2017).

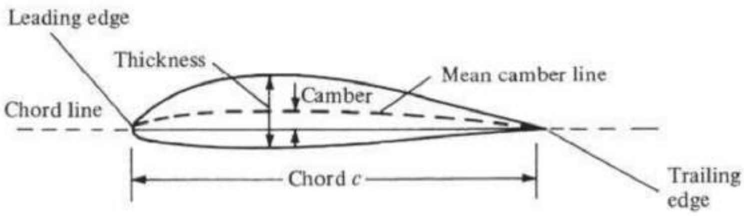



Figure 1: Airfoil geometric parameters. Adapted from: Anderson (2017).

As said before, the airfoil model that is going to be considered for the study and simulations is the NACA 0012, which is a symmetric airfoil, that is, the mean chamber line is equal to the chord. The main geometry problem consist in the airfoil and the angle of attack (AOA) $\alpha$ in which the airfoil is submitted (Fig. 2). In addition, there are the free steam velocity $U$ and the division between upper and lower surfaces, which were heated at constant temperature as wall boundary condition.

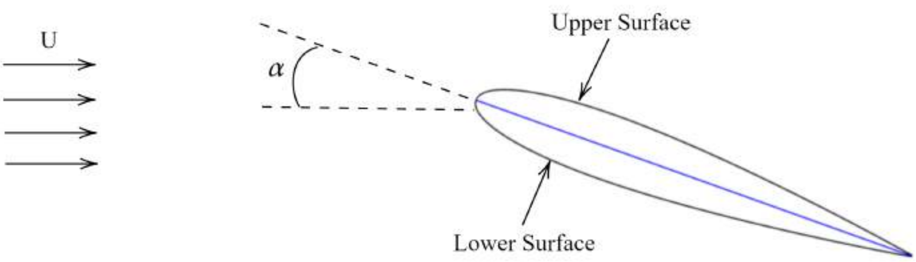



Figure 2: Airfoil Geometry

### 3.3 Turbulence Model

There are several turbulence models as investigated in Aftab *et al.* (2016) for low Reynolds airfoil flow, such as Direct Navier-Stokes (DNS), Spallart Allmaras (S-A), Reynolds Average Navier Stokes (RANS), SST K-$\omega$, standard K-$\epsilon$, realizable K-$\epsilon$ and so on. Some of the models cited above can be found in Versteeg and Malalasekera (2007).

Since the current work deals with low Reynolds Number ($R_e = 3000$), the turbulence model should have no significantly influence on the obtained results, considering what can be found on Çengel and Cimbala (2018), the turbulence is considered to happen from Reynolds number of higher magnitudes and is calculated according to equation 4. The realizable $k - \epsilon$ model was chosen since it is considered to have a better convergence behavior and more robust than standard $k - \epsilon$.

### 3.4 Simulation Configuration

As can be seen in Fig. 3, a C-type mesh was used to model the flow around the airfoil where $c$ has the value of 10 cm, proposed by Hassanaliana *et al.* (2019). This type of mesh geometry suits well for the construction of a structured mesh in the leading edge of the airfoil, eliminating problems of curvature associated to this part. The value of 25 times

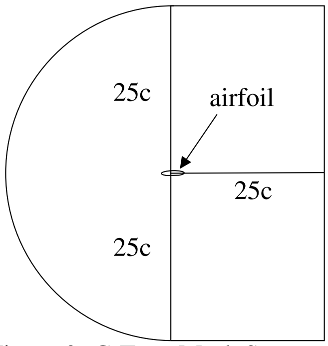



Figure 3: C-Type Mesh Structure

the chord is used based in related works such as Abobaker *et al.* (2020) and Ahmed *et al.* (2013b). This value has the main objective of obtain important effects on subsonic flow in the leading and trailing edge of the airfoil. As boundary conditions, the value of $Re = 3000$ is chosen based on related works Samiee *et al.* (2018). For the inlet condition, also, a condition of wall is used in the airfoil surface where different temperatures will be used in the upper and lower surface. In order to catch more influence of temperature gradient a temperature difference of 50 ºC between surfaces was used based on Samiee *et al.* (2018), so initially the airfoil heated at 350K at the upper surface and 300K at lower surface is simulated, then, the opposite is done. The resultant mesh can be found in Fig. 4 with 280000 elements.

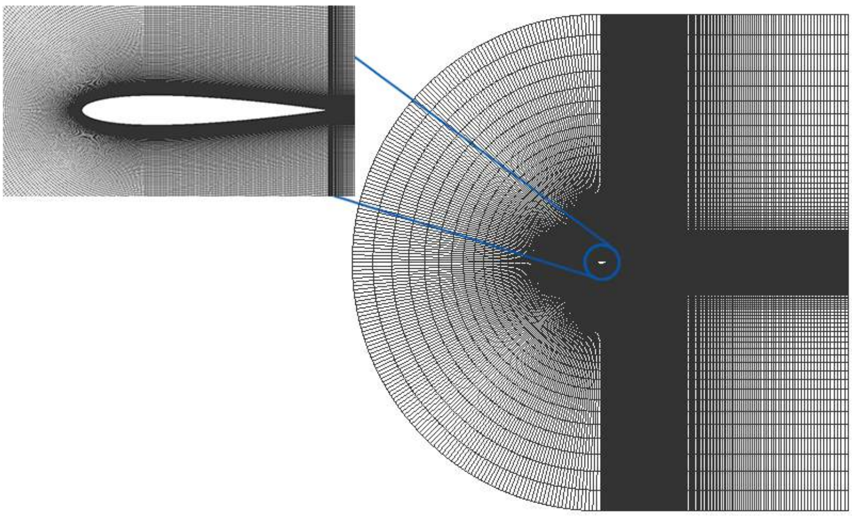


Figure 4: C-type Mesh around NACA 0012

As said before, the simulation was conducted using Ansys Fluent academic software. The air was considered as an ideal gas and the Sutherland Law is used in order to model the viscosity, since this flow parameter varies with temperature gradients. The Reynolds Number is given by the following equation

$$Re = \frac{\rho V c}{\mu} \tag{4}$$

where $\rho$ is the fluid density, V is the flow velocity, c is the chord length and $\mu$ is the dynamic viscosity. Since in this work we are dealing with $Re = 3000$, for the air properties at 298 K, the flow velocity is $V = 0.4686 m/s$, this velocity was used for the inlet boundary condition. For this flow velocity, the fluid can be considered as incompressible flow (Çengel and Cimbala, 2018), so the solver was pressure-based and steady state. The scheme used was coupled, as default and the discretization was used as second order for all the properties in order to avoid numerical diffusion (Karadimou and Markatos, 2018). All the simulations were conducted at 1 atm as operation conditions.

After the simulation is done for each surface heating at 0º, 5º, 10º, 15º, 20º, 25º and 30º of AOA, the drag ($c_D$) and lift ($c_L$) coefficient is calculated, given by

$$c_D = \frac{D}{\frac{1}{2}\rho U^2 S}, \qquad c_L = \frac{L}{\frac{1}{2}\rho U^2 S}, \tag{5}$$

where $D$ and $L$ are the drag and lift force, respectively, and $S$ is the surface area, for a cord of $c = 0.1m$, $S = 0.1m^2$.

After the boundary conditions were defined, what is obtained in terms of temperature influence over the flow for an AOA of $15°$ can be seen on Fig. (5).

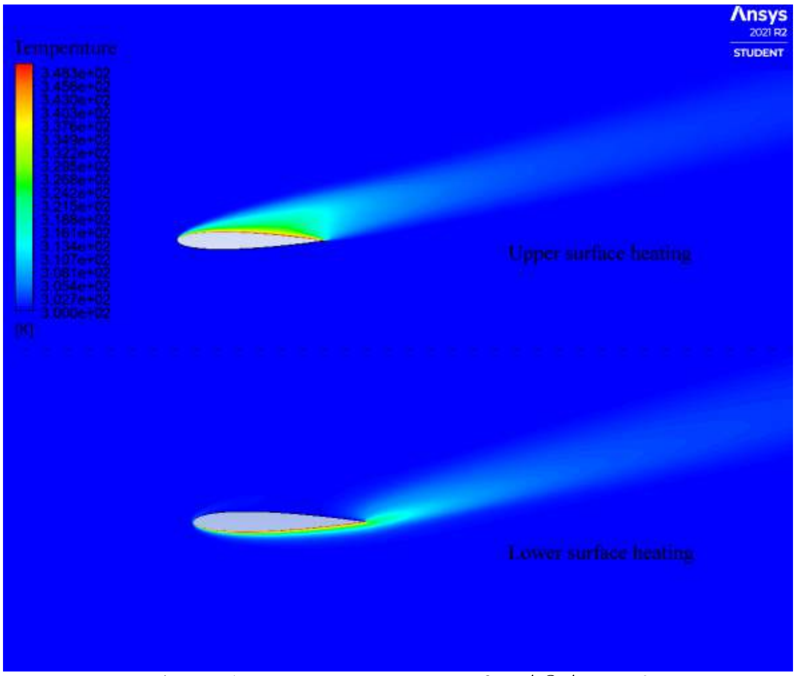


Figure 5: Temperature contour for $AOA = 15º$

### 3.5 Mesh Independence Study

The accuracy of the numerical solution in the physical domain depends on both the error related to the solution at the grid points and the error of interpolation. There are multiple reasons for the error arising on numerical simulation. First one, the mathematical model do not represent well the physical problem. Second, the mathematical model arise a numerical error at some stage of the simulation. Third, the numerical error is influenced by the size and shape of the grid cell (mesh). Fourth, there is an error based on the interpolation of the cells (Liseikin, 2017). In order to prevent some of the problems cited above with the mesh, an independence study was conducted with numerous values of element quantity in the mesh and the value of $c_L$ and $c_D$ was evaluated. For the simulation, an angle of attack of 5º was chosen and all the other conditions were the same for the final mesh simulations. The result can be seen in Fig. 6.

According to Fig. 6, after around 200k elements there is a small difference between $c_L$ and $c_D$, so for the final mesh a value of 280k elements was selected (Fig. 4).

## 4. Results

As can be seen in Fig. (7a), increasing the upper surface temperature is actually harmful for the lift coefficient in comparison to not heating. Very similar analogy can be made for the drag coefficient, presented in Fig. (7b).

Considering the lower surface, the results are far more desirable in terms of engineering and aerodynamics. The first thing to notice is that the lift coefficient for $0°$ presents a slightly greater value when compared to the airfoil with no heating and even greater when compared to the upper surface warmed. Another parameter to be observed is that the behaviour is maintained up until an angle of approximately $10°$. From this angle on, there's a convergence between the

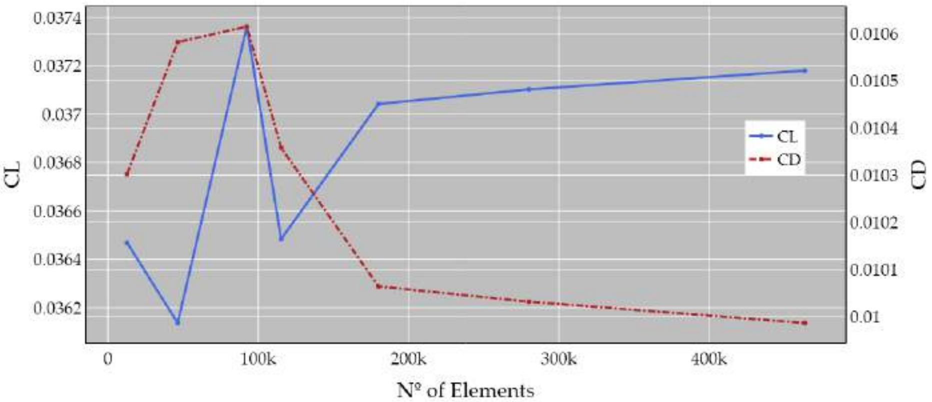
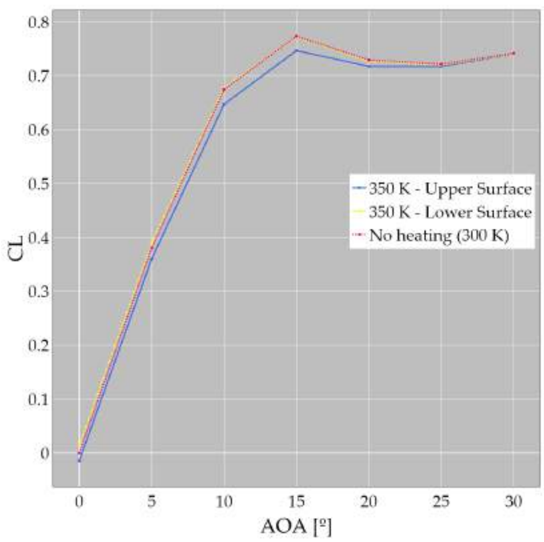



Figure 6: Mesh refinement influence over aerodynamic coefficients

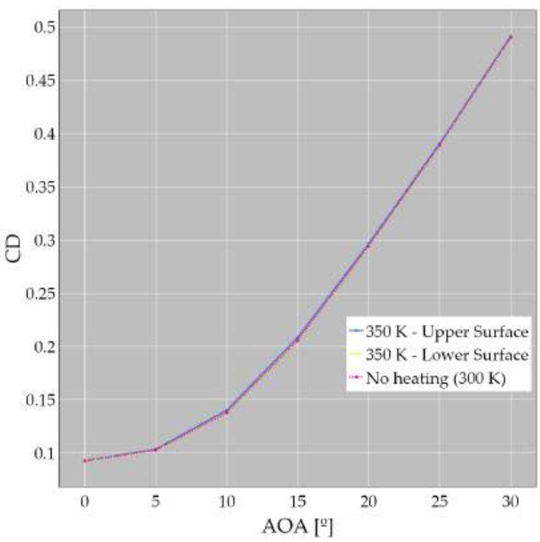
model with lower surface heated and with no heat applied to the airfoil. Just as for the previous case, a similar analogy can be viewed for the drag coefficient as well, mainly when compared to the upper surface heated.

It also can be observed on Fig. (7a) a different behaviour than the one that is normally encountered on the graphics of $c_L vs. \alpha$, once that the stall angle (abrupt lift loss) is not really important for this airfoil. The explanation for that relies mainly on the fact that the operation Reynolds value is too low for this abrupt loss to happen.

On the other hand, the curve of the Fig. (7b) of $c_D vs. \alpha$ shows a completely expected behaviour enhancing with the increasing of $\alpha$, once that as the angle of attack keeps getting bigger, the surface area in contact with the free stream is also increasing and, therefore, the drag produced too.




(a) Graphic obtained for the lift coefficient *vs.* AOA ($\alpha$).




(b) Graphic obtained for the drag coefficient *vs.* AOA ($\alpha$).

Figure 7: Aerodynamic Coefficients *vs.* AOA ($\alpha$).

Though it might seen little, when the aerodynamic coefficients are evaluated together on the lift-to-drag ratio (Fig. 8), an important parameter for these studies, we can infer that the upper surface heated presents unsatisfactory performance even when compared to the standard airfoil evaluation. Also, for $\alpha < 10$, it can be seen that the lower surface heated shows better performance for the NACA 0012 airfoil case. After this value, a convergence can be observed between standard case and lower surface heated case, this reveals a low influence of temperature difference for high angle of attack. A temperature contour can be seen in Fig. 5, this show the influence of heating one of the surface on the temperature profile.

As a final remarks, it is known that increasing temperature decrease density and this affects directly the value of lift coefficient since it depends of this fluid parameter (Anderson, 2017), so as expected, there is a reduction of $c_L$ at some AOA. In contrast, when the fluid is heated the viscosity increase (Çengel and Ghajar, 2015), this returns a bigger value of $c_D$ and this can be noted in the results (Tab. 1).

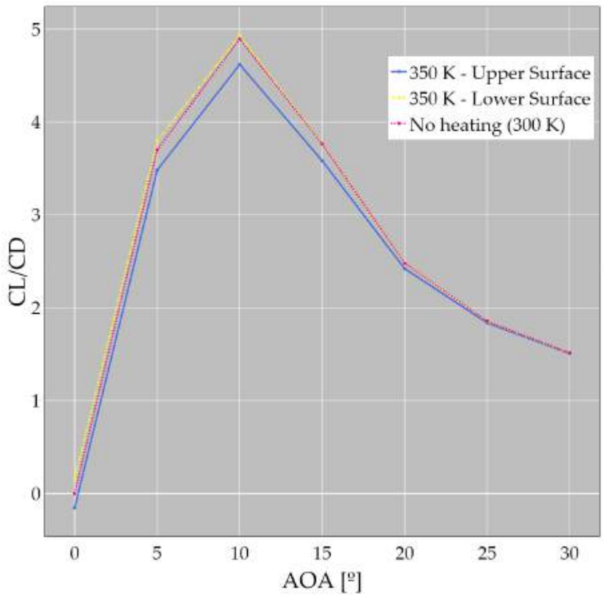



Figure 8: Graphic obtained for the lift-to-drag ratio *vs.* AOA ($\alpha$).

| | Upper (350 K) | Upper (350 K) | Lower (350K) | Lower (350K) |
|---|---|---|---|---|
| | $c_L$ (%) | $c_D$ (%) | $c_L$ (%) | $c_D$ (%) |
| 0º | -1,44 | 7,18E-03 | 1,44E+00 | 7,18E-03 |
| 5º | -1,96 | 7,68E-02 | 9,29E-01 | -2,06E-02 |
| 10º | -2,76 | 2,26E-01 | 3,37E-01 | -3,43E-02 |
| 15º | -2,65 | 2,88E-01 | -1,48E-01 | -3,90E-02 |
| 20º | -1,17 | 2,24E-01 | -2,07E-01 | -3,03E-02 |
| 25º | -4,13E-01 | 1,70E-01 | -1,63E-01 | -2,87E-02 |
| 30º | -1,65E-01 | 1,37E-01 | -1,58E-01 | -3,72E-02 |

Table 1: Difference between heating and no heating surface results.

## 5. Conclusion

As the main conclusion for this paper, the authors have shown that if heat is going to be applied on the studied airfoil NACA 0012, it should be applied on the lower surface. Considering the objectives of the study, there are two assumptions. The first one is about the thermal investigation that could be properly realized and observed via CFD methods. The second one it's needed to look into the dimensionless parameters to realize that, even for these low Reynolds range, heating the lower surface would increase the performance of the airfoil as a feasible way to optimize the flight, once that the optimizations on lift and drag coefficient would happen if the heating system did not change the aerodynamic of the vehicle and if the heating system is light enough in order not to be so costly in terms of energy needed for the operation of the vehicle.

As a preliminary study, the presented results are interesting and further investigation considering biomimetic inspired problems should be done. For next studies, it would be interesting to check on other airfoil with a similar study, preferably non symmetrical. Another aspect that could be evaluated is the threshold value for the Reynolds number that the thermal influence can be measured and its effects aren't negligible compared to the flow. In addition, another study that could be made is comparing several airfoil to see if a pattern can be found due to their symmetry or if each case must be studied individually.

## 6. References

Abobaker, M., M.Elfaghi, A. and Addeep, S., 2020. "Numerical study of wind-tunnel wall effects on lift and drag characteristics of naca 0012 airfoil". *Akademia Baru: CFD Letters*, Vol. 12. doi:https://doi.org/10.37934/cfdl.12.11.7282.

Aftab, S., Mohd Rafie, A., Razak, N. and Ahmad, K., 2016. "Turbulence model selection for low reynolds number flows". *PLoS ONE*, Vol. 11.

Ahmed, M.T., Amin, S.T., Islam, R. and Ahmed, S., 2013a. "Computational study of flow around a naca 0012 wing flapped at different flap angles with varying mach numbers". *Global Journal of Researches in Engineering General Engineering*, Vol. 13. ISSN 2249-4596.

Ahmed, T., Amin, M.T., Islam, S.R. and Ahmed, S., 2013b. "Computational study of flow around a naca 0012 wing flapped at different flap angles with varying mach numbers". *Global Journal of Researches in Engineering General Engineering*, Vol. 13.

Anderson, J., 1995. *Computational Fluid Dynamics*. McGraw-Hill Education, New York, 1st edition.

Anderson, J., 2017. *Fundamentals of Aerodynamics*. McGraw Hill Education, New York, 6th edition.

Benyus, J.M., 1997. "Biomimicry: Innovation inspired by nature".

Blazek, J., 2015. *Computational Fluid Dynamics. Principle and Applications*. Elsevier, Oxford, 3rd edition.

Bottlender, P.H.M., dos Santos, C.G., Demarco, G. and da Silva Benoit, A.G.M., 2021. "Biomimetics insetion in engineering: a literature review". *COBEM 2021*, Vol. 26.

Eftekhari, S. and Al-Obaidi, A., 2019. "Investigation of a naca0012 finite wing aerodynamics at low reynold's numbers and 0° to 90° angle of attack". *J Aerosp Technol Manag*. doi:10.5028/jatm.v11.1023.

Hassanaliana, M., Abdelmoula, H., Ayed, S.B. and Abdelkefi, A., 2017. "Thermal impact of migrating birds' wing color on their flight performance: possibility of new generation of biologically inspired drones". *Journal of Thermal Biology*.

Hassanaliana, M., Pelleritoa, V., Sedaghatc, A., Sabric, F., Borvayehd, L. and Sadeghic, S., 2019. "Aerodynamics loads variations of wings with novel heating of top surface: Bioinspiration and experimental study". *Experiment Thermal and Fluid Science*, Vol. 109.

Karadimou, D.P. and Markatos, N.C., 2018. "Study of the numerical diffusion in computational calculations". In S.P. Rao, ed., *Numerical Simulations in Engineering and Science*, IntechOpen, Rijeka, chapter 4. doi: 10.5772/intechopen.75660.

Liseikin, V.D., 2017. *Grid Generation Methods*. Springer, Glasgow, 3rd edition.

Samiee, A., Djavareshkian, M.H., Feshalami, B.F. and Esmaeilifar, E., 2018. "Improvement of airfoils aerodynamic efficiency by thermal camber phenomenon at low reynolds number." *J Aerosp Technol Manag*, Vol. 10. doi: 10.5028/jatm.v10.954.

Versteeg, H. and Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics. The Finite Volume Methods.* Pearson, Glasgow, 2nd edition.

Vincent, J.F.V., 2009. "Biomimetics – a review". *Proceedings of the Institution of Mechanical Engineers Part H Journal of Engineering in Medicine*. doi:10.1243/09544119JEIM561.

Çengel, Y. and Ghajar, A., 2015. *Heat and Mass Transfer. Fundamentals and Applications.* McGraw Hill Education, New York, 5th edition.

Çengel, Y.A. and Cimbala, J.M., 2018. *Fluid Mechanics. Fundamentals and Applications*. McGrill Hill Education, New York, 4th edition.

## 7. RESPONSIBILITY FOR THE INFORMATION

The authors are the only responsible for the included information of this work.