XL CILAMCE
IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGINEERING
NOVEMBER 11-14, 2019
Praiamar Natal Hotel & Convention
Natal, RN-BRAZIL

A COMPUTATIONAL STUDY OF THE HEAT TRANSFER PROCESS IN ANNULAR FINS

Lucas E. F. de Souza
Jonatha W. S. Araújo
Sandi I. S. de Souza
lucaseduardo@ufrn.edu.br
jonatha@ufrn.edu.br
sandi@ufrn.br
Universidade Federal do Rio Grande do Norte-UFRN
Av. Sen. Salgado Filho 3000 – Candelária, 59064-741, Natal - RN, Brazil

Abstract. In this work numerical simulation techniques was used as a way to evaluate the effects of natural convection on a horizontal tube with annular fins. Three configurations were studied by varying the spacing between the fins, 5 mm, 10 mm and 20 mm. The changes in the process of heat transfer as a function of the spacing S and the existence of natural driving were investigated. For this, a geometric model with two domains was developed. A domain contains fluid, Air outside, and a solid domain composed of a steel tube with fins, where a fixed temperature has been set at the base of the fins. A geometric model was developed and a CFD commercial code, ANSYS-CFX, was used in the numerical solution of the transport equations. Results for the fields of temperature in the fins and the heat transfer rates are presented. The results showed that the spacing between the fins has a thermal influence until stabilized. The temperature distribution on the face were similar.

Keywords: Convective Heat Transfer, Finned tube, Numerical Simulation.

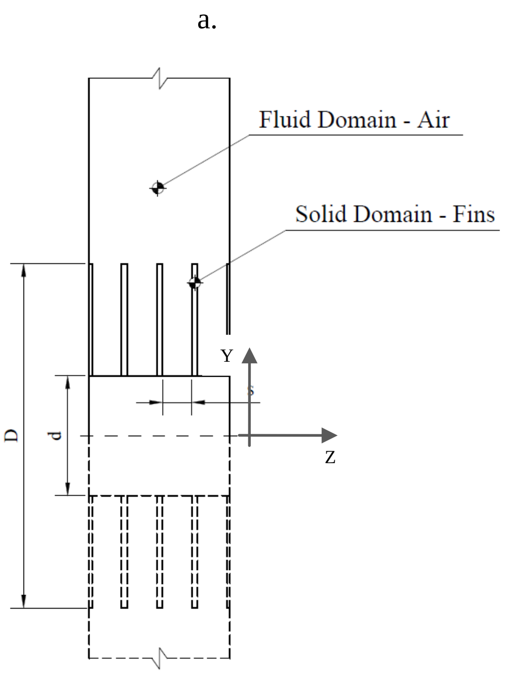
1 Introduction

In heat transfer, the devices are used to exchange energy between fluids, called heat exchangers. To improve the efficiency of these systems, fins are used, which improve thermal performance and decrease the size and weight of the equipment [3], are generally manufactured from metallic materials that have higher coefficient of thermal conduction. We can consider examples that use this type of technology, as in air conditioning, refrigerator or other type of heat exchange applications. These exchangers are generally compact, with a dense matrix of finned tubes and at least one of the flowing fluid [2].

In our daily lives, we can see heat-exchanging appliances, in the refrigerating there is an intense use of heaters and coolers where the use of fins is of vital importance. Electronic equipment like computers also employ fins to maximize heat exchanges and protect their components. Thus, the need for research aimed at improving the efficiency of the thermal exchange process is demonstrated [4] [5]. Research that uses experimental techniques in this area has a high cost, since a physical model is necessary, as well as to measure the variables of interest in various positions. A viable tool to study this class of problems with lower cost and with good results is the numerical simulation, where it is possible to develop realistic computational models in a flexible way, being easily adaptable to several types of physical conditions. The computational fluid dynamics (CFD) technique allows the development of options and projects, which can be analyzed and tested, allowing the development of an ideal project [6].

Thus, the objective of this work is the numerical study of the influence of fin spacing using the constant temperature condition on the wall. For this, a geometric model was developed with two distinct domains, steel and air, solid and fluid domains respectively. To solve the transport equations a commercial code in CFD, ANSYS-CFX, was used.

2 Physical model, computational mesh and boundary conditions



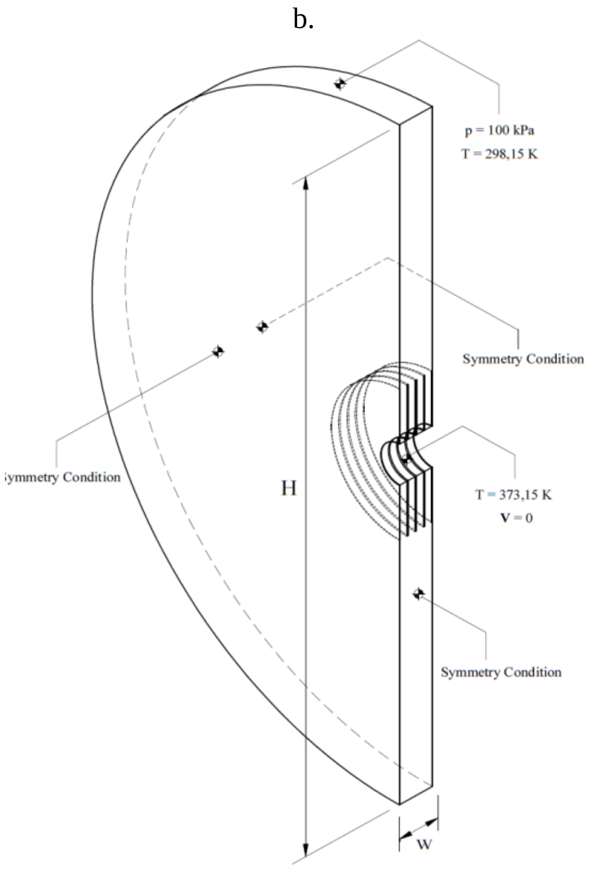


Figure 1. a. Computational model developed for simulations b. Boundary conditions

The Figure 1a shows the domain used in the study, it contains a fluid domain (air) and a solid domain (fins). To reduce computational cost a symmetry condition in the half plane of the model was applied.

Table 1 - Main dimensions of the model used

Dimensions	Value [m]
S	0.005, 0.01 e 0.02
D	0.100
d	0.028
δ	0.001
H	0.560

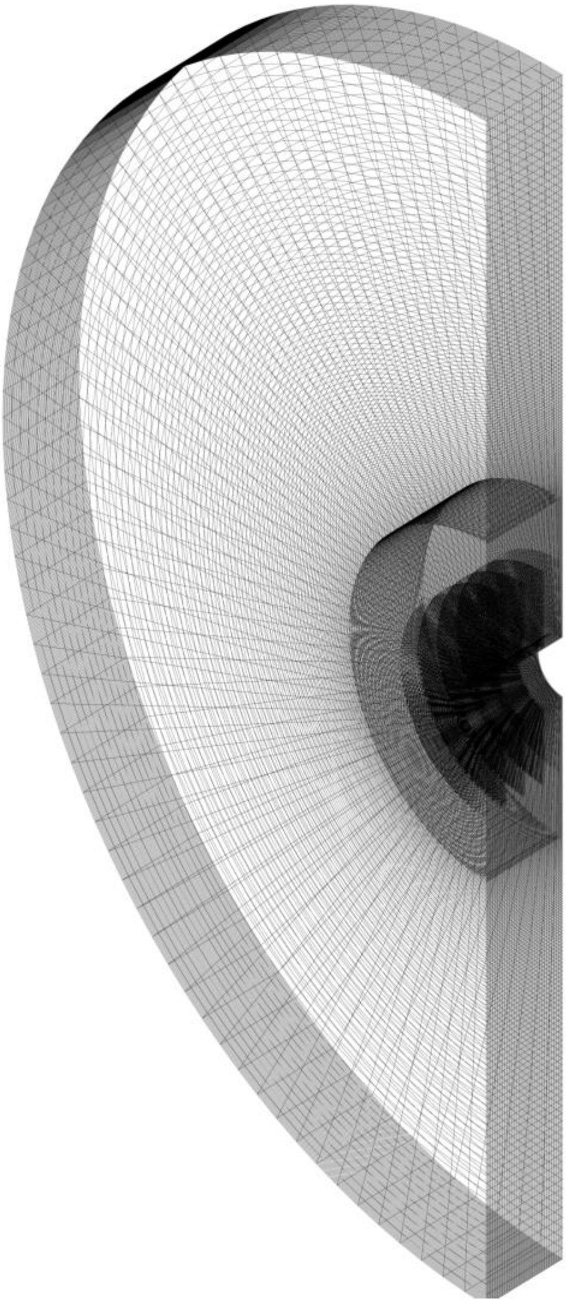


Figure 2: Mesh in all domains

Figure 2 shows the structured mesh used, where in the largest simulated case the mesh has 5.2 million elements and its orthogonal quality is around 0.999. Table 1 shows the dimensions of the models used, the variables used are described in Figure 1, where δ is the thickness of the fin.

The Figure 1b shows the boundary conditions employed in the numerical simulation. A fixed temperature of 373,15 K was adopted on the inner wall of the tube-fin assembly. It was adopted on the external surface, the region of entry and exit of fluid in the air domain at a temperature of 298,15 K with constant pressure of 100 kPa. This surface was considered open. The material used for the fin was a steel with thermal conductivity indicated by the software. On the lateral and frontal surfaces it was evaluated as a symmetry contour. In this case, there is a fluid-solid interface integrating the air domain and the fin domain. Finally, grid independece tests were performed to verify the reliability of the results.

3 Governing equations

Steady state simulations were performed and a laminar flow regime was considered for the air domain. In order to modeling the air flow occurring in the outer zone, the equations of conservation of mass (continuity) and the Navier-Stokes equations, Eq. (1) and Eq. (2), respectively, are used. The heat transfer processes were modeled by the energy and thermal diffusion equations, Eq. (3) and Eq. (4), respectively. They were solved using commercial software, ANSYS-CFX. The convergence criterion adopted was a mean square residual (RMS) of 10-6 for all variables involved.

$$\frac{\partial}{\partial x_j}\left(\rho U_j\right)=0 \tag{1}$$

$$\frac{\partial}{\partial x_j}\left(\rho U_i U_j\right)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\right]+S_M . \tag{2}$$

$$\frac{\partial}{\partial x_j}\left(\rho U_j T\right)=\frac{\partial}{\partial x_j}\left(\frac{k_f}{c_p}\frac{\partial T}{\partial x_j}\right) . \tag{3}$$

$$\frac{\partial}{\partial x_j}\left(k_s\frac{\partial T}{\partial x_j}\right)=0 . \tag{4}$$

In the equations above ρ is the specific mass of the fluid, μ the dynamic viscosity, x represents the specific coordinates, U is the velocity vector, p is the pressure, k and S_M represent the thermal diffusion coefficient and the source term respectively. T is the temperature and c_p the specific heat at constant pressure.

4 Results

Figure 3 shows the velocity distribution between two fins, in a line created 32 mm from the base of the tube. With the increase of the spacing between the fins a significant elevation with the air flow that penetrates between them, causing with that the convective coefficient increases and thus having an improvement in the efficiency and the heat flow. For the smaller spacing, comparing with S = 20 mm, the diffusive effects are more dominant, knowing that the velocity in this region is low. With the S = 5 mm, in all the data collection is present the boundary layer, for the other cases the maximum speed and end of the boundary layer are apparent, but a variation of the space where it ends, concluding that with increasing the spacing to a reduction in the boundary layer area.

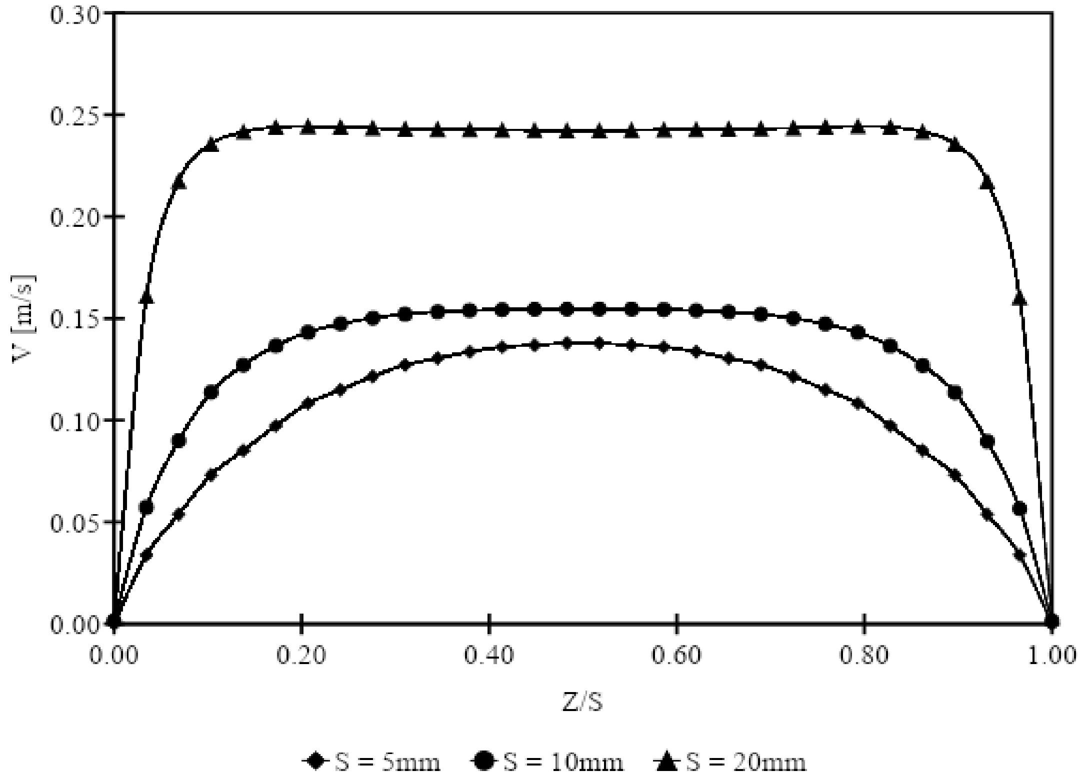


Figure 3: Velocity distribution between fins - line positioned at a radial distance of 32 mm

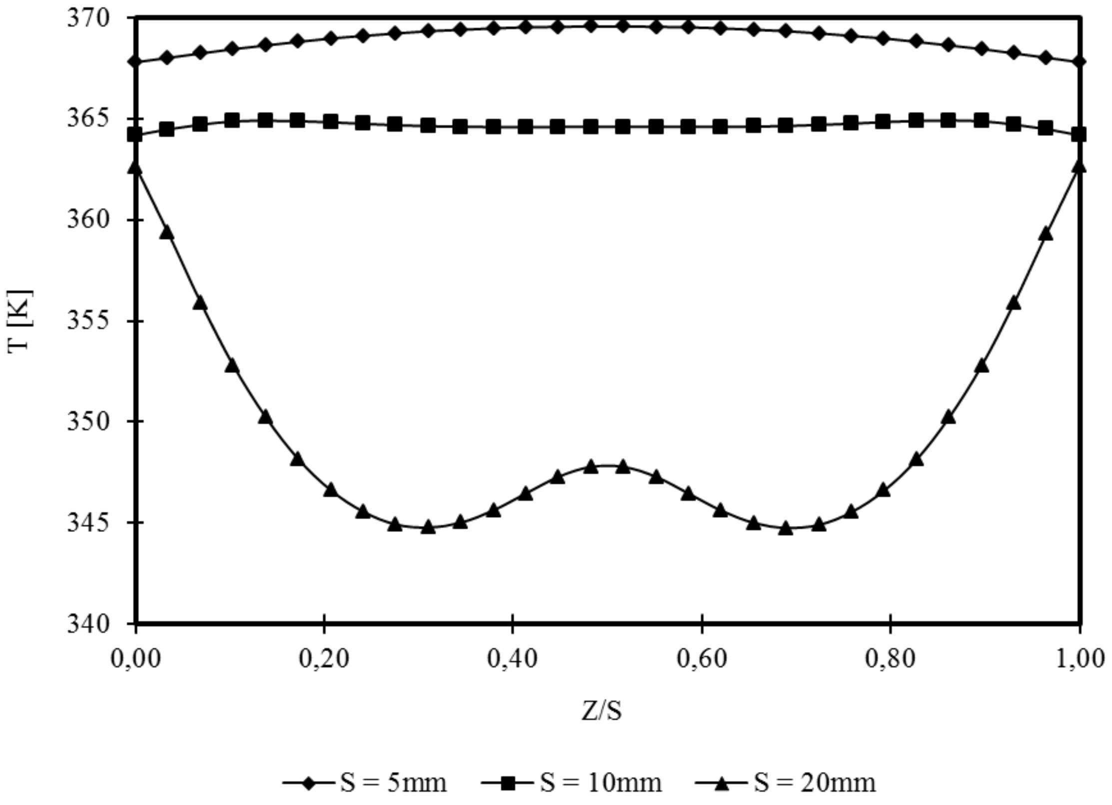


Figure 4: Temperature distribution between fins - line positioned at a radial distance of 32 mm from the Z - axis

Figure 4 shows the temperature distribution on the same radial line as Figure 3. For cases with spacings S = 5 mm and S = 10 mm there is a short variation temperature distribution in the interior between the fins, because air velocities in both These cases are low, thus hinders the heat convection mechanism in this region and making the diffusive mechanism predominate. For the case of S = 20 mm it is already noticed that the air temperature in the central region between the fins is considerably lower than in the cases of 5 and 10 mm, this is justified by the greater space for air circulation and consequently a higher speed in this region making the convection mechanism predominant.

.

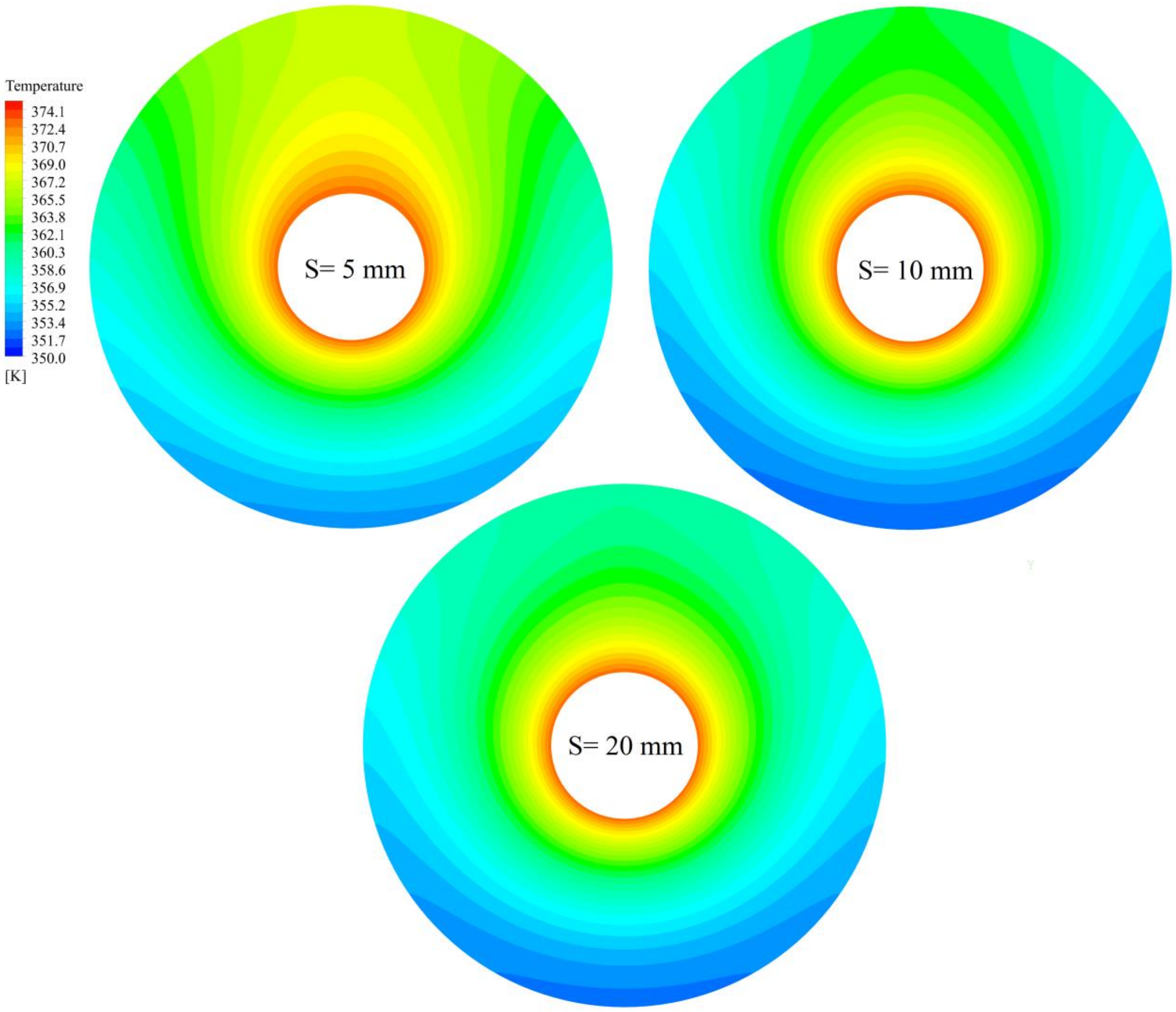


Figure 5: Temperature distribution at fins surface

Figure 5 presents the temperature distribution on the central fin of the model for each case studied. On the surface of the fins a maximum temperature variation of approximately $\Delta T = 24$ K was obtained. As the spacing S is increased A reduction in surface temperature values is observed, due to the increase in air flow between the fins, which makes the process of heat transfer in the fins greater.

Table 2 - Heat transfer rate in the central fin

S [mm]	Q_T [W]	Q_p [W]	ε [%]
5	1,288	0,238	18,47826
10	3,333	0,228	6,840684
20	3,471	0,206	5,934889

Table 2 shows the heat transfer values on the central fin, where Q_T is the total heat that the fin transfers to the outside, Q_p is the heat on the fin tip and ε is the influence of the heat transfer from the tip with all the fin. It is observed that with the increase in spacing there was an increase in heat transfer, as already observed, but at S = 10 mm and S = 20 mm the difference is very small compared to S = 5 mm and S = 10 mm, which may make a hypothesis that the spacing will be so large that it will no longer influence this thermal property, to conclude this is necessary simulate more cases, leaving a hint for further research. It is noticeable that the variation of the heat flux at the tip of the fin remains almost constant, having a slight decrease. As the spacing decreases, the fin tip becomes considerably valid in the thermal analysis of this type of equipment, knowing that in some calculations for this type of study is given as despicable.

Conclusions

In this work a computational model was developed for a case study of the fin heat transfer through CFD simulation. three cases were developed S = 5 mm, S = 10 mm and S = 20 mm. The work consists of a preliminary analysis, however, the consistency of the showsthe importance and ability of simulation processes to represent reality. In future work a larger amount of simulations will be performed in order to obtain stronger conclusions.

Acknowledgements

Firstly, God allowed all of this to happen, for he is the greatest teacher that anyone can know. UFRN, for the availability of space in the CFD laboratory and necessary support.

References

[1] Chen, H. T. and Hsu, W. L. 2007. Estimation of heat transfer coefficient on the fin of annular-fined tube heat exchangers in natural convection for various fin spacings. *Int. J. Heat and Mass Transfer* 50:1750-1761.

[2] Incropera, F. P. D. P. and DeWitt, T.L. Bergman, and A.S. Lavine. 2015. *Fundamentals of heat and mass transfer*, 7th ed. John Wiley & Sons, New York.

[3] Bhuyan, A. A.; AMIN, M, R.; ISLAM, A.K.M.S. "*Three-dimensional performance analysis ofplain fin tube heat exchangers in transitional regime*". Applied Thermal Engineering. p. 445-454, 2013.

[4] Kayansan, N. and Karabacak, R. 1992. Natural convection heat transfer coefficients for a horizontal cylinder with vertically attached circular fins. Heat Recovery Systems & CHP, 12(6):457-468.

[5] DOGAN, B and ALTUN, O; UGURLUBILEK, N; TOSUN, M; SARICAY, T. and ERBAY, L.B. *"Anexperimental comparison of two multi-louvered fin heat exchangers with different numbers of finrows".* Applied Thermal Engineering. p. 270–278, 2015.

[6] YAICI, W; GHORAB, M and GHORAB, E.; "*3D CFD analysis of the effect of inlet air flow maldistribution on the fluid flow and heat transfer performances of plate-fin-and-tube laminar heat exchangers*". International Journal of Heat and Mass Transfer 74, p. 490–500, 2014

[7] de Souza, S.I.S., de Bessa, K.L. & Maurente, A. J Braz. Soc. Mech. Sci. Eng. (2019) 41: 114. https://doi.org/10.1007/s40430-019-1609-y