

**ENC-2020-0029**

**CONSTRUCTAL THEORY APPLIED TO CONSTRUCTION OF FINS  
ARRANGEMENT MOUNTED IN A MICROCHANNEL SUBJECTED TO  
LAMINAR FORCED CONVECTIVE FLOWS**

**Rafael San Martin Moreira**

**Liércio André Isoldi**

**Jeferson Avila Souza**

**Elizaldo Domingues dos Santos**

Graduate Program of Ocean Engineering, School of Engineering, Federal University of Rio Grande - FURG, Rio Grande, Brazil,  
Italia Av., km 8, 96201-900.

rafaelsanmartin89@gmail.com

liercioisoldi@furg.br

jefersonsouza@furg.br

elizaldosantos@furg.br

**Abstract.** *Present numerical work investigates the construction of an arrangement of heated square fins mounted in a two-dimensional microchannel subjected to forced convective, incompressible and laminar cooled flows. The purpose is to maximize the heat transfer rate between the heated fins and fresh flow. Constructal Theory is employed for construction of the fins arrangement from an elemental configuration. The construction is based on a function that uses fluid dynamic and thermal fields for definition of each new elemental fin placement in the occupation area of the channel. For all simulated cases, forced convective flows with Bejan and Prandtl numbers of  $Be_L = 3.6$  and  $Pr = 0.71$  are investigated. The solution of conservation equations of mass, momentum and energy is performed with a numerical code based on the Finite Volume Method (FVM). The mounting of fins varied in the range  $1 \leq N \leq 15$ . For the present conditions, results indicated that an intermediate configuration with  $N = 4$  led to the highest heat transfer rate between fins and cooled flow in the channel. For  $N \geq 6$ , the fluid dynamic and thermal streams are strongly suppressed leading to high pressure drop and poor thermal exchange in the microchannels.*

**Keywords:** *Numerical Simulation, Geometrical Investigation, Constructal Theory, Microchannels, Heat Exchangers*

## 1. INTRODUCTION

Microchannel heat exchangers are devices used to obtain high magnitudes of heat exchange per unit volume. They are largely applied in aerospace, marine and automotive industry due to requirements of low weight and space. Recently, manufacturing costs have decreased spreading its employment in other fields as domestic cooling condensers, military, nuclear and electronic industry (Chen and Ding, 2011; Naqjuddin et al., 2018). According to Naqjuddin et al. (2018) the growth of electronic industry depends on the solution of heat generation problem and warranty of smooth operation of electronic devices, illustrating the importance of the microchannel heat exchangers.

One example of compact device is the Printed Circuit Heat Exchanger (PCHE), which have as main advantages the small volume and reduced weight in comparison with shell and tube or plate heat exchangers. Moreover, these devices support high pressure (due to excellent mechanical resistance at the interface of union) and elevated temperatures, attending wide operational conditions. For instance, according to Morteau (2017) these devices can support pressures over than 500 bar. Once the manufacturing of these heat exchangers allowed the manufacturing of even more complex designs and considering the importance of high heat exchange per unit volume in this kind of problem, the investigation of design becomes a key subject to improve even more the performance and compactness of these devices.

Considering the importance of microchannels in many engineering applications, several studies have been developed to investigate predefined geometric configurations of cross-sectional shapes of the channels or geometry of the mounted fins in the channel. Qu and Mudawar (2002) performed a numerical study solving laminar, forced convection water flows ( $Re = 1,400$ ) in microchannels with rectangular cross-sectional. The purpose was the comprehension of the distribution of temperature in the flow region and solid domain of the channel and validation of the developed computational method. The developed code was verified and validated by means of comparison with analytical and experimental results of literature. Bello-Ochende et al. (2007) performed a numerical study for geometrical optimization of three-dimensional heated microchannel subjected to forced convection heat transfer. The main purpose was the minimization of peak temperature in the microchannel walls. Constructal Design was employed for definition of constraints and geometric

ratios investigated for the predefined geometric configuration of the channel. The microchannel is shared in unity cells, which had a volume varying in the range  $0.1 \text{ mm}^3 \leq V \leq 0.9 \text{ mm}^3$  and pressure differences in the range  $10 \text{ kPa} \leq \Delta p \leq 75 \text{ kPa}$ . In this study it was investigated the length/height ratio of the unity cell of the microchannel. Results indicated that the studied ratio had a strong influence over the peak temperature in the domain. Kim et al. (2013) investigated the influence of cross-sectional shape of a PCHE over its thermal performance. More precisely, four different formats of cross-sectional channel (semicircular, rectangular, trapezoidal and circular) and three different performance parameters (thermal efficiency, heat exchange area and pressure drop) were investigated. The highest thermal efficiency was achieved for the rectangular channel, however it also led to the worst performance in terms of pressure drop. The semi-circular channels led to a similar thermal performance and with pressure drop significantly lower than that reached for rectangular cross-sectional shapes for the channels. In this sense, authors recommended the use of semi-circular cross-sectional channels. More recently, Feijó et al. (2018) employed Constructal Design and Exhaustive Search for maximization of heat exchange between two rectangular fins mounted in alternated surfaces of a channel and for minimization of pressure drop in the channel. For the thermal performance, results recommended the highest intrusion of the fins to maximize the heat transfer rate between the fins and surrounding flow, while the opposite configuration is the best for minimization of pressure drop. For the multi-objective investigation, the optimal upstream fin is taller than the downstream one, i.e., an asymmetric configuration for the fins.

Recent studies have focused in definition of arrangements of obstacles in channels without a predefined form, for instance, Daróczy et al. (2014) employed a heuristic method of optimization (Genetic Algorithm) to find numerically the best arrangement for placement of seven heated cylinders in a channel flow. Main purposes were the maximization of heat exchange between the heated tubes and minimization of pressure drop of the arrangement. In spite of important recommendations about the optimal design, the process of construction of the arrangement was not performed based on a physical principle. Moreover, only the final optimal configuration is the subject of the study. Recently, Pedrotti et al. (2020) studied numerically the construction of an arrangement of heated tubular array subjected to an external forced convective flow. The building of the arrangement was performed on the light of Constructal Design, positioning each tube in a deterministic way using a defined function dependent on the velocity and temperature fields. Moreover, four different minimum distances between the tubes and three different Reynolds numbers are studied. Results indicated that the optimal configurations were dependent of the minimum distance between the tubes and the Reynolds number, showing that the freedom given for the arrangement and the flow magnitude have strong influence over the generated design of the system.

It is worthy to mention that Constructal Theory is a viewing that the design of all finite flow systems (including those found in nature) is ruled by a physical principle named Constructal Law. Constructal Law states that the design of a finite flow system with freedom to morph along time will evolve in such way to improve the access to the internal currents that flow through it (Bejan, 2000; Bejan and Lorente, 2008; Bejan, 2016). The definition of a construction function performed in Pedrotti et al. (2020) and used in the present work is an attempt to define a method of growth of convective heat transfer flow systems based on the application of Constructal Theory, i.e., in a deterministic form. Here, it is performed a numerical study about the construction of an arrangement of heated square fins arrangement mounted in a region of the channel flow (occupation area) which is one of the problem constraints. The number of fins varied in the range  $1 \leq N \leq 15$ , being the first fin placed in a predefined position and the others are placed according to a construction function. It is considered a forced convective flow with  $Be_L = 3.6$  and  $Pr = 0.71$  for the 15 simulated cases. The solution of the conservation equations of mass, momentum and energy are solved with the Finite Volume Method (FVM) (Patankar, 1980; Versteeg and Malalasekera, 2007), more precisely with the commercial code FLUENT, version 14 (FLUENT, 2013).

## 2. MATHEMATICAL AND NUMERICAL MODELING

In the present study, it is considered a laminar, incompressible, steady and forced convective flow in a two-dimensional microchannel. It is also considered that there is no gravitational field, so natural convection is not being considered. The fluid motion is caused by the imposition of different pressures at the inlet and outlet of the channel in such way the Bejan number is  $Be_L = 3.6$  with null manometric pressure imposed at the channel exit. For this Bejan number a velocity magnitude of  $u = 0.05 \text{ m/s}$  is obtained at the channel inlet. The inferior and superior surfaces of the channel, as well as, each inserted fin have non-slip and impermeability boundary condition. Concerning the thermal conditions, at the inlet surface it is imposed a constant temperature of  $T_\infty = 300 \text{ K}$ , at the outlet surface it is considered an outflow boundary condition (locally parabolic) and for each mounted fin it is considered a constant temperature of  $T_S = 400 \text{ K}$ . It is worthy to mention that in the present modeling, the thermophysical properties are considered constant, not being affected in the regions near the fins. The air is considered as working fluid with a Prandtl number of  $Pr = 0.71$ . The channel has a height and length of  $H = 30 \text{ mm}$  and  $L = 190 \text{ mm}$ , respectively. With respect to the insertion of the fins, it is considered an occupation area of  $30 \text{ mm} \times 30 \text{ mm}$  where square fins of  $5 \text{ mm}$  of sides are mounted. The first fin has its placement predefined and the other fins follow a growth function defined as function of velocity and temperature fields. The center of the first fin is placed at the coordinates  $x_1 = 32.5 \text{ mm}$  and  $y_1 = 2.5 \text{ mm}$  (lower upstream region of occupation

area). Figure 1 illustrates the computational domain with imposed boundary conditions and dimensions of the problem, including the occupation area which is represented by the blank area.

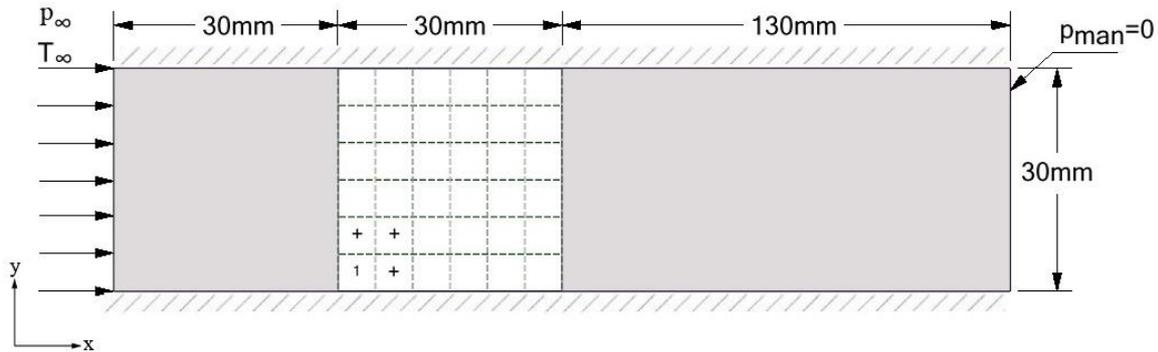


Figure 1 – Computational domain of the channel with occupation area where the fins are inserted.

The modeling of the present problem is represented by the conservation equations of mass, momentum in directions  $x$  and  $y$  and energy, which are given by (Bejan, 2013):

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (2)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (3)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) \quad (4)$$

where  $u$  and  $v$  are the velocities in  $x$  and  $y$  directions (m/s),  $x$  and  $y$  are the spatial coordinates (m),  $p$  is the pressure (N/m<sup>2</sup>),  $T$  is the temperature (K),  $\rho$  is the density of fluid (kg/m<sup>3</sup>),  $\nu$  is the kinematic viscosity (m<sup>2</sup>/s) and  $\alpha$  is the thermal diffusivity (m<sup>2</sup>/s).

In this work, the arrangement of fins is constructed instead of using a predefined configuration. The first configuration consists of a fin inserted in the lower surface of the cavity and each new fin is ruled by a construction function based on the fluid dynamic and thermal fields. Here it is performed an attempt to show how the design in flow systems can growths deterministically. Here a heat transfer problem is used to represent the flow system, but the same principle of design can be extrapolated for several flow systems by application of Constructal Law (Bejan, 2000; Bejan and Lorente, 2008; Bejan, 2016).

The purpose of the system is to maximize the heat transfer between the heated fins and the surrounding cooled flow per unit depth, which can be given by:

$$q' = \frac{hA\Delta T}{W} = \dot{m}'c(T_{out} - T_{\infty}) \quad (5)$$

where  $h$  is the convective heat transfer coefficient [W/(m<sup>2</sup>K)],  $A$  is the heat exchange area of the fins (m<sup>2</sup>),  $W$  is the depth of the channel (m),  $\dot{m}'$  is the mass flow rate per unit depth [kg/(ms)],  $c$  is the specific heat [J/(kgK)] and  $T_{out}$  is the mean temperature in the exit of the channel.

The heat transfer rate per unit length is dependent on the heat exchange area, the convective heat transfer coefficient and temperature difference. The area is augmented with the insertion of new fins, then it will not be considered in the construction function. Once the coefficient  $h$  for forced convective flows is dependent on the Reynolds number and gradient of temperature, the function established here is defined as a function of velocity and temperature fields by:

$$\varphi = \frac{U}{u_{\infty}} \left( \frac{T_f - T}{T_f - T_{\infty}} \right) \quad (6)$$

where  $U$  is the magnitude of velocity in any point of air flow in the channel domain (m/s),  $u_{\infty}$  is the velocity magnitude at the channel inlet (m/s),  $T_f$  is the fins temperature (K) and  $T$  is the temperature in any point of the air flow in the channel domain (K).

It is assumed here that the position where the variable  $\varphi$  is the maximum represents the point of highest potential for insertion of a new fin. However, the construction should respect the constraints established for the problem. Moreover, the fins can not be mounted in overlapped positions.

For construction of the arrangement, the same procedure employed in the work of Pedrotti et al. (2020) is used here. This construction consists on the definition of an algorithm to define the coordinates where the new inserted fin is placed. The Constructal algorithm is defined by the following steps:

- 1 – Define the construction function ( $\varphi$ );
- 2 – Define the dimensions of the occupation area ( $A_{oc}$ );
- 3 – Define the dimension of the fins (resolution of arrangement building) which are inserted in the occupation area;
- 4 – Define the position of the first square fin (here the center of the fin is placed at the coordinates  $x_I = 32.5$  mm and  $y_I = 2.5$  mm)
- 5 – Solve the fluid dynamic and thermal field of the problem;
- 6 – Calculate the scalar field of  $\varphi$  in the channel domain;
- 7 – Calculate the coordinate where  $\varphi$  is the maximum respecting a minimum distance between the fins;
- 8 – Generate a new domain with a new fin placed in the position where the highest  $\varphi$  is found;
- 9 – Return to step 5 while it is possible to insert new square fin, i.e., when the arrangement filled the occupation area.

Concerning the numerical modeling, all simulations are performed with software FLUENT version 14 (FLUENT, 2013) which is based on the Finite Volume Method (FVM) (Patankar, 1980; Versteeg and Malalasekera, 2007). It is employed rectangular volumes for discretization of the channel domain. For discretization of advective terms it is used the second order Upwind interpolation function. To tackle with the velocity-pressure coupling it is employed the SIMPLE scheme. Moreover, the solutions are considered converged when the residuals of mass, velocities in  $x$  and  $y$  directions and temperature is lower than  $10^{-6}$ .

The computational domain is spatially discretized in rectangular volumes. A study of grid sensibility is performed considering the case with one fin inserted in the channel. When new fins are intruded in the domain, similar discretization found for the first case is employed for the other fins. For achievement of independent grid, the number of volumes is varied and the spatial averaged convective heat transfer coefficient ( $h$ ) is monitored in the fin. When the percentual deviation between one mesh and the next refined mesh is  $dev < 0.1\%$  the mesh is considered independent. The parameter for achievement of independent grid is given by:

$$dev(\%) = 100 \frac{|h_{ave}^j - h_{ave}^{j+1}|}{h_{ave}^j} \quad (7)$$

where  $h_{ave}^j$  is the space averaged convective heat transfer coefficient for the course mesh and  $h_{ave}^{j+1}$  is the coefficient for the next refined mesh. Table 1 presents the results for the spatial averaged convective heat transfer coefficient as function of refinement. For the present study, it is considered the grid M3 as the independent. The same grid characteristics will be considered when the number of cases is varied.

Table 1. Results of grid sensibility study for the case with one inserted fin.

Grid	Size of Finite Volume (mm)	Number of Cells	$h_{ave}$	Dev (%)
M1	$2.0 \times 10^{-3}$	1,424	6.114	0.12
M2	$1.0 \times 10^{-3}$	5,676	6.107	0.11
M3	$5.0 \times 10^{-4}$	22,700	6.100	0.04
M4	$2.0 \times 10^{-4}$	141,876	6.098	-

To verify the present solution, it is simulated a problem with forced convective flow around a square bluff body with  $Re_B = 60$  and  $Pr = 1.0$  (where  $B$  is the bluff body sides) studied in the work of Sahu et al. (2009). The methodology used here is the same employed in the works of Feijó et al. (2018) and Teixeira et al. (2020) and the results obtained for verification case are the same presented in these works. In this sense, for the sake of brevity, the results of verification are not presented here and it can be seen in the above-mentioned works.

### 3. RESULTS AND DISCUSSION

In this section it is presented the results found for the construction of the arrangement of fins mounted in the channel subject to forced convective flow with  $Be_L = 3.6$  and  $Pr = 0.71$ . Initially, the temperature and pressure fields for some constructions are presented, showing the growth of the arrangement. Afterwards, as the purpose here is the maximization of the heat transfer between the fins arrangement and the surrounding flow, the energy on the inlet and exit of the channel is calculated to estimate the energy transfer from the fins to the air flow, and the effect of the number of fins over the heat transfer rate per unit length is presented.

Figure 2 illustrates the thermal and pressure fields, Fig. 2(a) and 2(b), for the predefined case, i.e., with the elemental fin inserted in the lower and upstream region of the occupation area. Figure 2(a) shows the thermal boundary layer generated by the presence of the fin, where the insertion of a new fin is not recommended due to the contact with a heated fluid stream. Then, in the thermal viewpoint, the new fin should be inserted in a region few affected by the thermal boundary layer, as expected. Figure 2(b) shows the pressure field for the same case with  $N = 1$ . As can be seen, the region where the fluid flow is expanded presented an intensification of the momentum, being an important potential region for insertion of a new fin. Results of pressure fields showed that regions near the first fin are not suitable for insertion of the second fin. The heat transfer rate per unit length found for this case is  $q' = 9.17$  W/m.

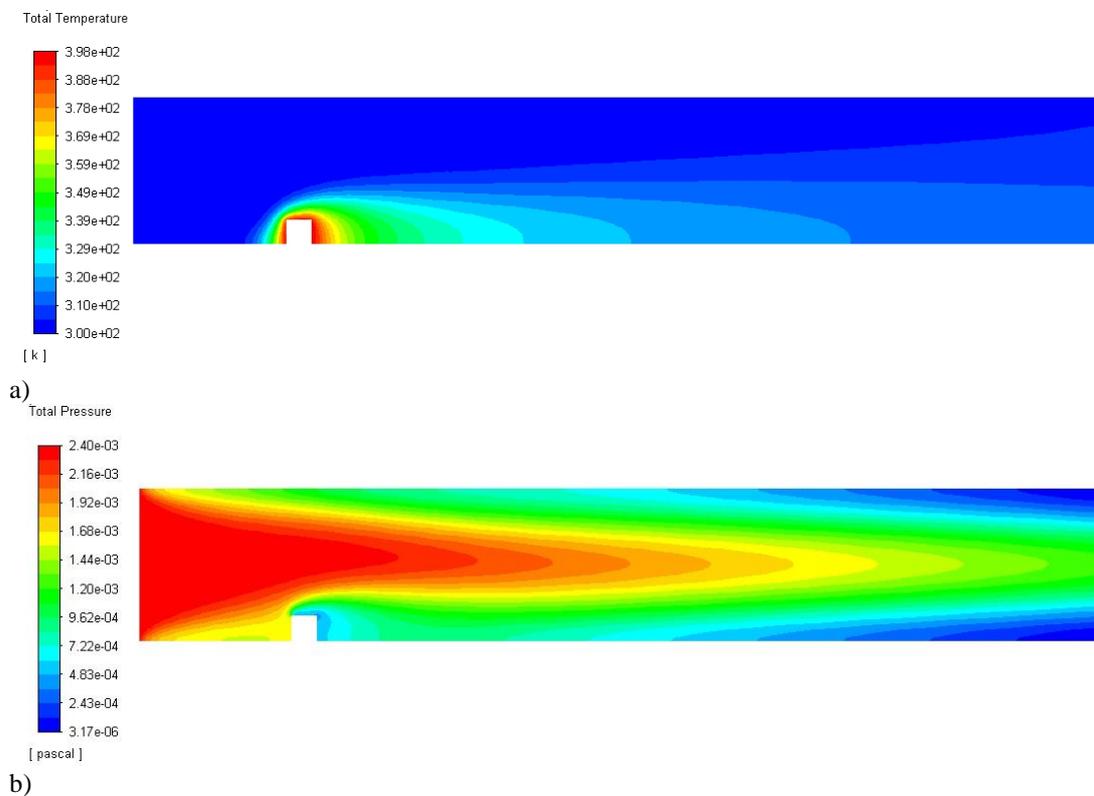


Figure 2. Fields obtained for the channel with one predefined fin ( $N = 1$ ) mounted in the domain: a) temperature, b) pressure.

Figure 3 shows the thermal and pressure fields, Fig. 3(a) and 3(b), for the configuration with  $N = 4$ , which represents the case with the highest heat transfer rate per unit length ( $q'_{max} = 23.39$  W/m). For  $N = 2$ , the highest construction function was found for the coordinates  $x_2 = 55$  mm and  $y_2 = 25$  mm, i.e., almost in the superior corner of the occupation area. The third fin displaces to a point near the first fin at the coordinates  $x_3 = 33.5$  mm and  $y_3 = 7.0$  mm and the fourth fin is placed slightly above the third fin ( $x_4 = 33.5$  mm and  $y_4 = 13.0$  mm). Results also presented for the thermal field a region between fins 2 and 4 where the stream is lower affected by heated blocks already mounted in the domain. Moreover, the region downstream the fins 1, 3 and 4 is not suitable for insertion of new fin due to the augmentation of the heated fluid region. The pressure field also indicated that the region between fins 2 and 4 is also the one with the lower resistance to fluid flow. Consequently, the fifth fin should be mounted in this region.

Figure 4 depicts the thermal and pressure fields, Fig. 4(a) and 4(b), for the arrangement with  $N = 6$ . From this configuration apart, the fins 1, 3, 4, 5 and 6 creates a wall that blocks the flow passage from the upstream to the downstream of the channel. Therefore, the fluid is heated significantly in downstream region of the channel and the insertion of new fins does not bring new contributions for heat transfer rate. Figure 4(b) shows that pressure field also has a dramatic drop in the fins arrangement corroborating the founding observed in Fig. 4(a). It is important to note that the

flow is laminar, with low flow intensity to prevent the flow from undergoing regime transition due to the insertion of obstacles.

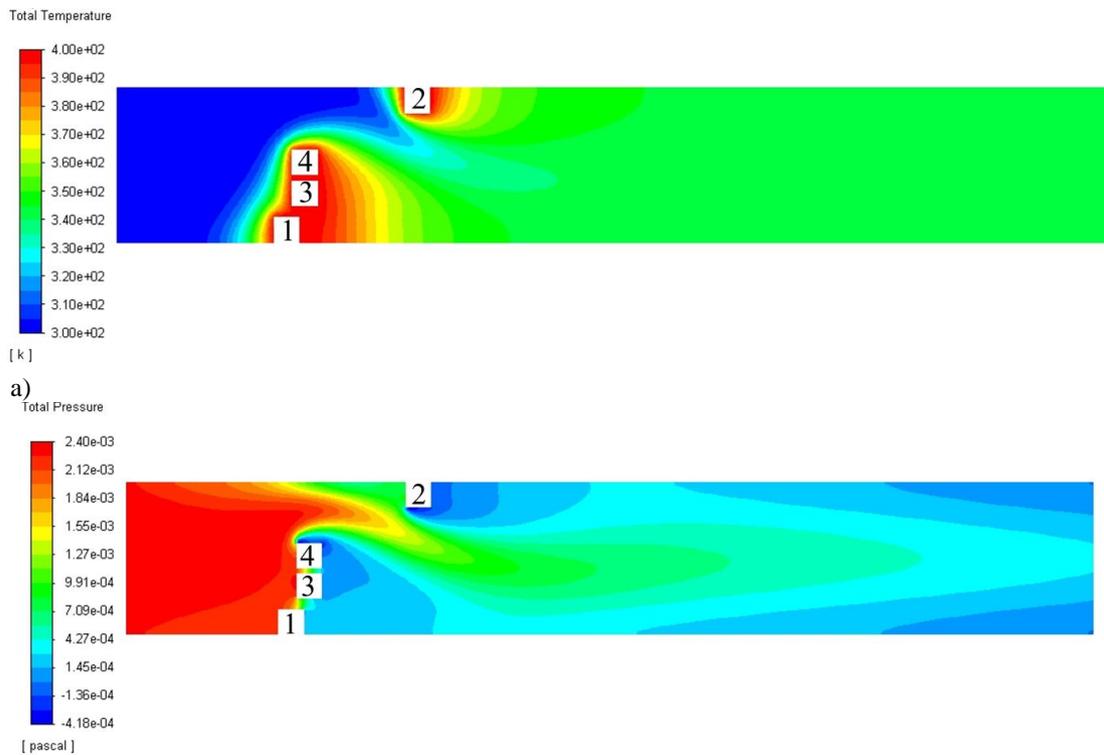


Figure 3. Fields obtained for the channel with the optimal arrangement ( $N = 4$ ) found in the present study: a) temperature, b) pressure.

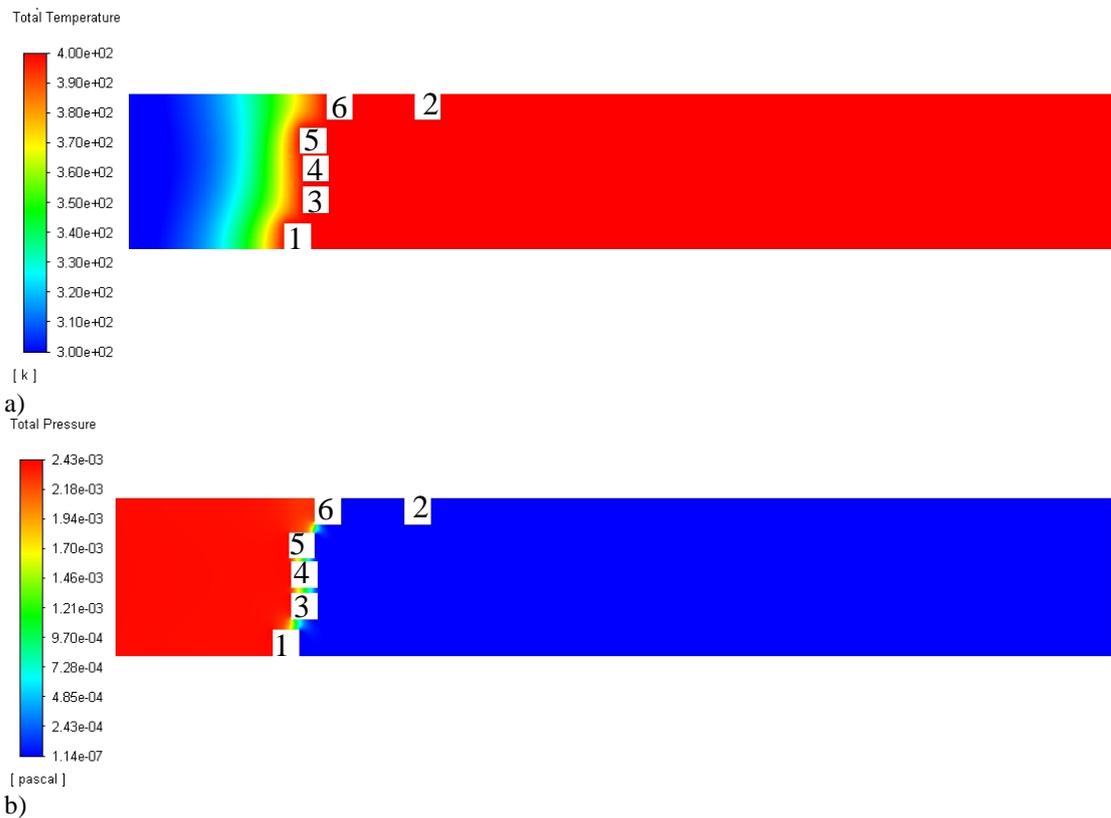


Figure 4. Fields obtained for the channel with  $N = 6$  found in the present study: a) temperature, b) pressure.

Figure 5 shows the temperature and pressure fields, Figs. 5(a) and 5(b), obtained for the case with the highest number of inserted fins ( $N = 15$ ). Results demonstrated that for  $N \geq 6$  the fins are placed in a region where the fluid is heated and the flow is restricted. Therefore, the insertion of new fins does not bring significant contribution to the heat transfer rate.

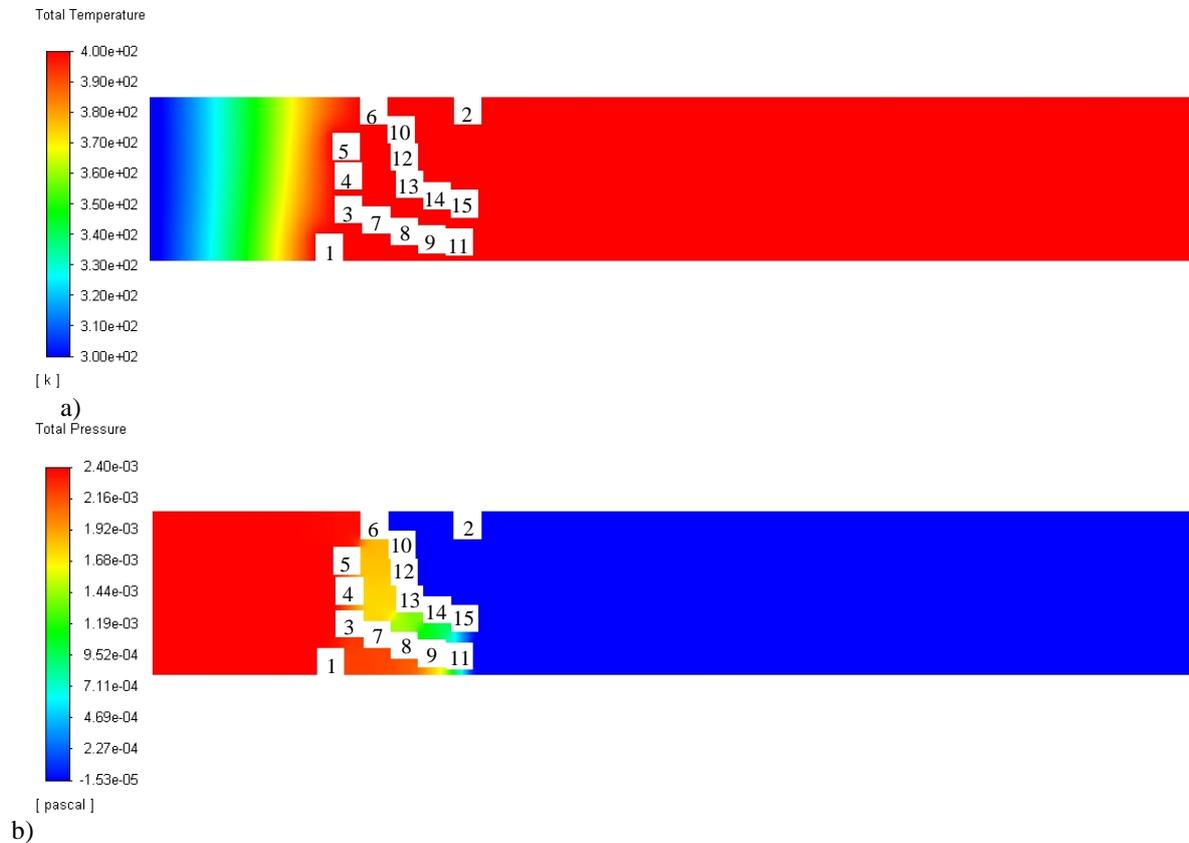


Figure 5. Fields obtained for the channel with  $N = 15$  found in the present study: a) temperature, b) pressure.

In order to show quantitatively the influence of the fins arrangement, Fig. 6 shows the effect of the number of fins over the heat transfer rate per unit length. It can be noticed an increase of the heat transfer rate per unit length from  $q' = 9.2$  W/m for  $N = 1$  to  $q'_{max} = 23.4$  W/m for  $N_o = 4$ , i.e., nearly 2.5 times higher. For values of  $N > 4$ , the magnitude of  $q'$  had a significant drop in its magnitude up to  $N = 6$ , where the decrease changed to a smooth form, indicating the saturation of occupation area. The arrangement with the highest number of fins ( $N = 15$ ) led to the worst performance ( $q' = 2.8$  W/m), which is 8.4 times inferior than the best configuration.

In general, results indicated the importance of design for this kind of problem since the best configuration presented a significant superior performance than the worst configuration. Moreover, results demonstrated that for the present thermal conditions the most complex shape is not suitable. Probably, the simulation of higher magnitudes of  $Be_L$  can lead to the optimal configuration with a higher number of inserted fins. The best shape found presented an irregular configuration, which is a pattern commonly found in natural systems, mainly for those subjected to various performance indicator. In this sense, there is an indicative that the employment of a construction function is a promise technique to predict the formation of a convective heat transfer arrangement.

#### 4. CONCLUSION REMARKS

The present numerical work investigated the employment of a Constructal Law algorithm to build the arrangement of heated fins inserted in a channel subjected to forced convective and laminar flows. The main goal is to obtain the geometric configuration for the arrangement of fins that led to the best thermal performance of the problem, as well as, identify the evolution of the design mounting the fins one by one using a Construction function based on fluid dynamic and thermal fields. For all simulated cases, it is considered a forced convective flow with  $Be_L = 3.6$  and  $Pr = 0.71$ .

Results indicated that the employment of a construction function led to promising results, achieving an irregular optimal configuration similar to that noticed in some natural systems. Moreover, an intermediate complex configuration ( $N_o = 4$ ) led to the best performance with  $q'_{max} = 23.4$  W/m, which was 2.5 and 8.4 times superior than the predefined configuration with  $N = 1$  and the most complex shape with  $N = 15$ , respectively. The simplest configuration ( $N = 1$ ) had

a small heat exchange area, while the complex configurations ( $N \geq 6$ ) imposed a strong restriction to fluid dynamic and thermal streams, as noticed in pressure and temperature fields. In other words, the technique not only was used as a powerful tool to improve the thermal performance of the system as demonstrated that the augmentation of complexity does not necessarily led to the best performance. One possible reason for this behavior is concerned with the intensity of the forced convective system. Based in previous findings obtained in Pedrotti et al. (2020) for arrangement of ducts in a channel flow, probably the simulation of higher magnitudes of  $Be_L$  can conduct to the generation of more complex configurations.

Future studies must be performed to improve the comprehension about the application of the Constructal algorithm. For instance, queries about the influence of the size of the elemental Constructal (side of the fins), the sequence of construction (one by one, two by two), intensity of fluid flow (varying the Bejan number), and even the Construction function employed (using pressure instead of velocity) are worthy of investigation.

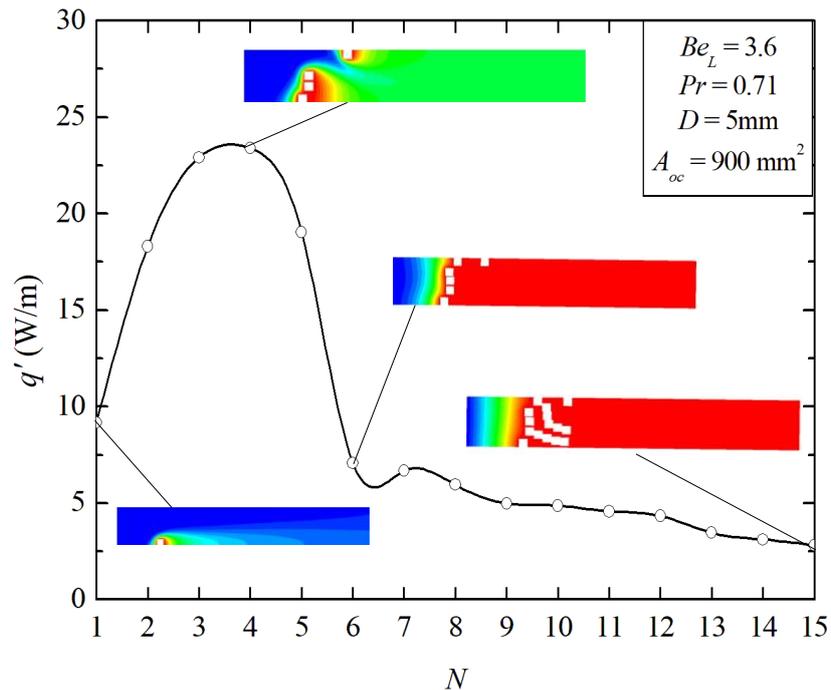


Figure 6. Effect of the number of fins ( $N$ ) over the heat transfer rate per unit length ( $q'$ )

## 5. ACKNOWLEDGEMENTS

L. A. Isoldi, J. A. Souza and E. D. dos Santos thank CNPq for research grant (Processes: 306012/2017-0, 306024/2017-9, 306734/2016-8). All authors thank FAPERGS for financial support in PqG Program – Notice N° 05/2019 (Process: 19/2551-0001847-9).

## 6. REFERENCES

- Bejan, A. and Lorente, S., 2008. *Design with Constructal Theory*. John Wiley & Sons, Hoboken, New Jersey, USA.
- Bejan, A., 2000. *Shape and structure, from engineering to nature*. Cambridge University Press, New York, USA.
- Bejan, A., 2013. *Convection Heat Transfer*, John Wiley & Sons, New York, USA.
- Bejan, A., 2016. *The physics of life: the evolution of everything*. St. Martins Press, New York, USA.
- Bello-Ochende, T., Liebenberg, L. and Meyer, J.P., 2007. “Constructal cooling channels for micro-channel heat sinks”. *International Journal of Heat and Mass Transfer*, Vol. 50, 4141 – 4150.
- Chen, C. and Ding, C., 2011. “Study on the thermal behaviour and cooling performance of a nanofluid-cooled microchannel heat sink”. *International Journal of Thermal Sciences*, Vol. 50, pp. 378 – 384.
- Daróczy, L., Janiga, G. and Thévenin, D., 2014. “Systematic analysis of the heat exchanger arrangement problem using multi-objective genetic optimization”. *Energy*, Vol. 65, pp. 364 – 373.

- Feijó, B.C., Lorenzini, G., Isoldi, L.A., Rocha, L.A.O., Goulart, J.N.V. and Dos Santos, E.D., 2018. “Constructal design of forced convective flows in channels with two alternated rectangular heated bodies”. *International Journal of Heat and Mass Transfer*, Vol. 125, pp. 710 – 721.
- FLUENT, 2013. *Documentation Manual* – FLUENT 14.3.
- Kim, S.M. and Mudawar, I., 2013. “Universal approach to predicting heat transfer”. *International Journal of Heat and Mass Transfer*, Vol. 56, n 1 – 2, pp. 238 – 250.
- Mortean, M.V.V., 2017. *Trocadores de calor compactos soldados por difusão: fabricação e modelagem*. Ph.D. Thesis, Universidade Federal de Santa Catarina, Florianópolis, Brasil.
- Naqiuddin, N.H., Saw, L.H., Yew, M.C., Yusof, F. and Ng, M.K., 2018. “Overview of micro-channel design for high heat flux application”. *Renewable Sustainable Energy Reviews*, Vol. 82, pp. 901 – 914.
- Patankar, S.V., 1980. *Numerical heat transfer and fluid flow*. McGraw Hill, New York, USA.
- Pedrotti, V.A., De Escobar, C.C., Dos Santos, E.D., Souza, J.A., 2020. “Thermal analysis of tubular arrangements submitted to external flow using constructal theory”. *International Communications in Heat and Mass Transfer*, Vol. 111, pp. 104458-1 – 104458-12.
- Qu, W. and Mudawar, I., 2002. “Analysis of three-dimensional heat transfer in micro-channel heat sinks”. *International Journal of Heat and Mass Transfer*, Vol. 45, pp. 3973 – 3985.
- Sahu, A.K., Chhabra, R.P., and Eswaran V., 2009. Effects of Reynolds and Prandtl Numbers on Heat Transfer from a Square Cylinder in the Unsteady Flow Regime. *International Journal of Heat and Mass Transfer*, Vol. 52, pp. 839–850.
- Teixeira, F.B., Altnetter, M.V., Lorenzini, G., Rodriguez, B.D.A., Rocha, L.A.O., Isoldi, L.A. and Dos Santos, E.D., 2020. Geometrical Evaluation of a Channel with Alternated Mounted Blocks under Mixed Convection Laminar Flows Using Constructal Design. *Journal of Engineering Thermophysics*, Vol. 29, n. 1, 92 – 113.
- Versteeg, H.K. and Malalasekera, W., 2007. *An introduction to computational fluid dynamics: the finite volume method*. Pearson Education Limited.

## 7. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.